3rd Polish Congress of Mechanics

21st International Conference on Computer Methods in Mechanics

Editors
Michał Kleiber
Tadeusz Burczyński
Krzysztof Wilde
Jarosław Górski
Karol Winkelmann
Łukasz Smakosz
3rd Polish Congress of Mechanics

and

21st International Conference
on
Computer Methods in Mechanics

Short Papers
Vol. 1

Editors:
Michał Kleiber    Jarosław Górski
Tadeusz Burczyński    Karol Winkelmann
Krzysztof Wilde    Łukasz Smakosz

Gdańsk 2015
Organizers
Polish Society of Theoretical and Applied Mechanics
Polish Association for Computational Mechanics
Institute of Fundamental Technological Research of the Polish Academy of Sciences
Committee on Mechanics of the Polish Academy of Sciences
Section of Computational Methods and Optimization
Committee on Civil Engineering and Hydroengineering of the Polish Academy of Sciences
Section of Mechanics of Structures and Materials
Committee on Machine Building of the Polish Academy of Sciences
Institute of Fluid-Flow Machinery of the Polish Academy of Sciences
Gdańsk University of Technology
Faculty of Civil and Environmental Engineering
Department of Structural Mechanics

Chirmanship
Congress President
Michał Kleiber (Polish Academy of Sciences)

Congress Vice-President
Włodzimierz Kurnik (Warsaw University of Technology)

Chairman of the Scientific Committee
Tadeusz Burczyński
(Institute of Fundamental Technological Research of Polish Academy of Sciences)

Vice-Chairman of the Scientific Committee
Krzysztof Wilde (Gdańsk University of Technology)

Chairman of the Permanent Congress Committee
Arkadiusz Mężyk (Polish Society of Theoretical and Applied Mechanics)

Honorary Chairman
Witold Gutkowski (Polish Academy of Sciences)

Honorary Patronage
Minister of Science & Higher Education of the Republic of Poland
Marshal of the Pomorskie Voivodeship
Rector of Gdańsk University of Technology
Director of Institute of Fluid-Flow Machinery Polish Academy of Sciences

Sponsors
Ministry of Science and Higher Education (MNiSW), Poland
ENERGA Group
SOFiSTiK AG, Oberschleißheim, Germany

Media Patronage
Acta Energetica
Honorary Committee

Romuald Będziński, Zielona Góra
Czesław Cempel, Poznań
Krzysztof Dems, Łódź
Andrzej Garstecki, Poznań
Józef Giergiel, Rzeszów
Witold Gutkowski, Warszawa
Zbigniew Kączkowski, Warszawa
Józef Kubik, Bydgoszcz
Krzysztof Marchelek, Szczecin

Jarosław Mikielewicz, Gdańsk
Zenon Mróz, Warszawa
Janusz Orkisz, Kraków
Andrzej Styczek, Warszawa
Gwidon Szefer, Kraków
Eugeniusz Śwoiński, Gliwice
Andrzej Tylkowski, Warszawa
Zenon Waszczyszyn, Kraków
Edmund Wittbrodt, Gdańsk

Scientific Committee

Krzysztof Arczewski, Warszawa
Jan Awrejcewicz, Łódź
Janusz Badur, Gdańsk
Czesław Bajer, Warszawa
Stefan Berczyński, Szczecin
Wojciech Blajer, Radom
Roman Bogacz, Warszawa
Ryszard Buczkowski, Szczecin
Tadeusz Burczyński, Warszawa
Witold Cecot, Kraków
Wojciech Cholewa, Gliwice
Jacek Chróścielewski, Gdańsk
Czesław Cichoń, Kielce
Paweł Dłużeński, Warszawa
Piotr Doerffer, Gdańsk
Stanisław Drobiak, Częstochowa
Dariusz Gawin, Łódź
Józef Gawlik, Kraków
Wojciech Gilewski, Warszawa
Zbigniew Gronostajski, Wrocław
Jan Holnicki-Szulc, Warszawa
Krzysztof Kalisziński, Gdańsk
Tomasz Kapitaniak, Łódź
Jan Kiciński, Gdańsk
Marian Klasztorny, Warszawa
Michał Kleiber, Warszawa
Paweł Klosowski, Gdańsk
Zbigniew Kołański, Łódź
Piotr Konderla, Wrocław
Witold Kosiński, Warszawa (†)
Janusz Kowal, Kraków
Tomasz Kowalewski, Warszawa
Zbigniew Kowalski, Warszawa
Katarzyna Kowal-Michalska, Łódź
Ireneusz Krecia, Gdańsk
Tomasz Krzyżyński, Koszalin
Mieczysław Kuczma, Poznań
Włodzimierz Kurkiewicz, Warszawa
Tomasz Lewiński, Warszawa
Tadeusz Łagoda, Opole
Wojciech Pietraszkiewicz, Gdańsk
Maciej Pietrzyk, Kraków
Stanisław Radkowski, Warszawa
Wojciech Radomski, Warszawa
Jacek Rokicki, Warszawa
Błażej Skoczewski, Kraków
Stanisław Stupkiewicz, Warszawa
Andrzej Jacek Teichman, Gdańsk
Jerzy Warmiński, Lublin
Krzysztof Wilde, Gdańsk

International Advisory Board

Jorge Ambrosio, Portugal
Klaus Jürgen Bathe, USA
Jian-Shyan Chen, USA
Rene de Borst, The Netherlands
Leszek Demkowicz, USA
Jüri Engelbrecht, Estonia
Marc Geers, The Netherlands
Dietmar Gross, Germany
Francois Jouvet, France
Reinhold Kienzler, Germany
Rimantas Kacianauskas, Lithuania
Pierre Ladeveze, France
Jolanta Lewandowska, France
Janos Logo, Hungary
Giulio Maier, Italy
Herbert Mang, Austria
Eugenio Onate, Spain
Manolis Papadrakakis, Greece
Ekkehard Ramm, Germany
Franz Remmers, Austria
Bernhard Schrefler, Italy
Paul Steinmann, Germany
Joao Antonio Teixeira de Freitas, Portugal
Hisaaki Tobushi, Japan
Viggo Tvergaard, Denmark
Wolfgang Wall, Germany
Organizing Committee

Gdańsk University of Technology
Faculty of Civil and Environmental Engineering
Department of Structural Mechanics

Chairman
Jarosław Górski

Vice-Chairmen
Paweł Kłosowski
Jacek Pozorski - Institute of Fluid-Flow Machinery PAS

Secretaries
Karol Winkelmann
Łukasz Smakosz

Members
Karol Daszkiewicz
Violetta Konopińska
Alina Krzycała
Marcin Kujawa
Jacek Lachowicz
Aleksandra Mariak
Anna Mleczek
Magdalena Rucka
Agnieszka Sabik
Marek Skowronenk
Mateusz Sondej
Katarzyna Szepietowska
Wojciech Witkowski
Beata Zima

Address
Gdańsk University of Technology
Faculty of Civil and Environmental Engineering
Department of Structural Mechanics
Narutowicza 11/12
80-233 Gdańsk, Poland
pcm-cmm-2015@pg.gda.pl
Phone: +48-58-347-21-74
Preface

This book brings us a great honour and pleasure to present the papers selected for presentation at the PCM-CMM-2015 CONGRESS held on 8-11 September, 2015 in Gdansk (Poland).

The PCM-CMM-2015 CONGRESS is a joint scientific event of the 3rd Polish Congress of Mechanics (PCM) and the 21st International Conference on Computer Methods in Mechanics (CMM).

The idea of a Polish Congress of Mechanics was firstly suggested in 2005 by the Polish Society of Theoretical and Applied Mechanics. The scope was intended to cover the whole range of problems of theoretical, experimental and computational mechanics as well as interdisciplinary issues, including industrial applications.

The 21st International Conference on Computer Methods in Mechanics continues the 44-year series of conferences dedicated to numerical methods and their applications to the mechanics-based problems. The meetings, organized biannually since 1973 provide a forum for presentation and discussion of new ideas referring to the theoretical background and practical applications of computational mechanics.

Both events – the 3rd Polish Congress of Mechanics (PCM) and the 21st Conference on Computer Methods in Mechanics (CMM) – are aimed at presenting current state-of-the-art research in the field of mechanics and providing a wide forum for discussion of new ideas on theoretical background, new technologies and computational methods in a vast domain of mechanics and related disciplines. We believe that the Congress becomes an event to trigger discussions, exchange of new ideas and valuable solutions in numerous aspects of mechanics. We hope that the book dedicated to academics, researchers, designers and engineers dealing with various problems of mechanics will take appropriate interest and meet a broad readers’ response.

Each paper submitted to PCM-CMM-2015 CONGRESS and printed in the book has been reviewed by members of the Scientific Committee and the International Advisory Board and refined by the Authors according to the referee comments. We are deeply indebted to all members the SC and IAB for their help in shaping the programme of the Conference and their important contribution to the publishing process of the book volumes. Most of the final texts have been additionally adjusted to technical requirements of the publisher, the English of some texts has been refined too. We would like to send our words of gratitude to our associates: Marek Skowronek, Marcin Kujawa, Anna Mleczek, Karol Daszkiewicz, Beata Zima, Aleksandra Mariak and Jacek Lachowicz for their assistance and help in bringing the volume to its final form.

Tadeusz Burczyński
Chairman of the Scientific Committee
# Table of contents

## Plenary lectures

<table>
<thead>
<tr>
<th>Speaker</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>J. Ambrósio</td>
<td>Interactions between mechanical systems and continuaum mechanical models in the framework of biomechanics and vehicle dynamics</td>
<td>1</td>
</tr>
<tr>
<td>M. Geers, V. Kouznetsova, A. Sridhar, A. Krushynska</td>
<td>Multiscale mechanics of dynamical metamaterials</td>
<td>7</td>
</tr>
<tr>
<td>R. Kienzler, P. Schneider</td>
<td>Consistent plate theories – A matter still not settled?</td>
<td>9</td>
</tr>
<tr>
<td>T.A. Kowalewski, P. Nakielni, F. Pierini, K. Zembrzycki, S. Pawłowska</td>
<td>Nanoscale challenges of fluid mechanics</td>
<td>11</td>
</tr>
<tr>
<td>Z.L. Kowalewski</td>
<td>Experimental attempts for creep and fatigue damage analysis of materials – state of the art and new challenges</td>
<td>17</td>
</tr>
<tr>
<td>M. Kuczma</td>
<td>Shape memory materials and structures: modelling and computational challenges</td>
<td>23</td>
</tr>
<tr>
<td>T. Kuretyka</td>
<td>Advanced mechanics in High Energy Physics experiments</td>
<td>25</td>
</tr>
<tr>
<td>B. Oesterle, M. Bischoff, E. Ramm</td>
<td>Hierarchic isogeometric analyses of beams and shells</td>
<td>27</td>
</tr>
<tr>
<td>W. Rachowicz</td>
<td>Finite Element Method simulations of linear and non-linear elasticity problems with error control and mesh adaptation</td>
<td>29</td>
</tr>
<tr>
<td>A. Soldati</td>
<td>Physics and computations of turbulent dispersed flows: macro - consequences from micro - interactions</td>
<td>31</td>
</tr>
<tr>
<td>V. Tvergaard</td>
<td>Finite strain analyses of deformations in polymer specimens</td>
<td>33</td>
</tr>
</tbody>
</table>

## Mini-symposia

### MS01 organized by: W. Cecot, W. Rachowicz, G. Zboiński

#### Adaptive Methods and Error Estimation

<table>
<thead>
<tr>
<th>Speaker</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>M. Abramowicz</td>
<td>The application of model parameter estimation method to detect connection damage in a steel-concrete beam using modal force residuals</td>
<td>35</td>
</tr>
<tr>
<td>W. Cecot, M. Oleksy, M. Krówczyński</td>
<td>Study of convergence of the multigrid homogenization</td>
<td>37</td>
</tr>
<tr>
<td>M. Klimczak, W. Cecot</td>
<td>Integration of hp-adaptive FEM and local numerical homogenization</td>
<td>39</td>
</tr>
<tr>
<td>J. Kucwaj</td>
<td>The influence of different equivalent boundary conditions on approximate solution to a potential problem</td>
<td>41</td>
</tr>
<tr>
<td>Ł. Miazio, G. Zboiński</td>
<td>Stress convergence in adaptive resolution of boundary layers in the case of 3D-based first- and higher-order shell models</td>
<td>43</td>
</tr>
<tr>
<td>O. Ostapov, O. Vovk, H. Shynkarenko</td>
<td>Computable double-sided a posteriori error estimates for h-adaptive Finite Element Method</td>
<td>45</td>
</tr>
<tr>
<td>J. Ptaszny</td>
<td>A fast multipole Boundary Element Method in the analysis of 3D linear elastic structures</td>
<td>47</td>
</tr>
<tr>
<td>W. Rachowicz, A. Zdunek, W. Cecot</td>
<td>An adaptive Finite Element Method for contact problems in finite elasticity</td>
<td>49</td>
</tr>
<tr>
<td>V. Stelemashchuk, H. Shynkarenko</td>
<td>Numerical modeling of thermopiezoelectricity steady state forced vibrations problem using adaptive Finite Element Method</td>
<td>51</td>
</tr>
<tr>
<td>G. Zboiński</td>
<td>Application of the element residual methods to dielectric and piezoelectric problems</td>
<td>53</td>
</tr>
</tbody>
</table>
MS02 organized by: Y. Vetyukov, M. Krommer
Axially Moving Structures

M. Baumgart, A. Steinboeck, M. Saxinger, A. Kugi
Elasto-plastic bending of steel strip in a hot-dip galvanizing line ................................................................. 55

P.G. Gruber, Y. Vetyukov, M. Krommer
Plastic deformation of axially moving continuum in mixed Eulerian-Lagrangian formulation ........................................ 57

A. Humer, L. Vu-Quoc, I. Steinbrecher
Complete modeling of the dynamics of sliding beams with large deformation .......................................................... 59

D. Ritzberger, A. Schirrer, S. Jakubek
Formulating the perfectly matched layer as a control optimization problem .......................................................... 61

J. Rusin
Vibrations of a double-beam complex system subjected to a moving force ............................................................ 63

E. Thonhofer, S. Jakubek
Online parameter identification for traffic simulation via Eulerian and Lagrangian sensing ........................................ 65

Y. Vetyukov, P.G. Gruber, M. Krommer
Modeling finite deformations of an axially moving elastic plate with a mixed Eulerian-Lagrangian kinematic description .... 67

MS03 organized by: J. Pamin, J. Tejchman, A. Winnicki
Computational Mechanics of Concrete and Geomaterials

P. Alawdin, A.I. Mordich, J.A. Muzychkin
Experimental and numerical analysis of precast-monolithic building floors under in-plane loading ................................ 69

J. Bobiński, J. Tejchman
Simulations of cracks in concrete with gradual transition from continuous to discontinuous description ............................. 71

K. Cichocki, M. Ruchwa
Distribution of damage in unconventionally reinforced concrete slabs subjected to impact loads .................................. 73

W. Grymin, M. Koniorczyk, D. Gawin
Mathematical model of concrete degradation due to the alkali-silica reaction at the mesoscopic level .............................. 75

I. Jankowiak
XFEM analysis of intermediate crack debonding of FRP strengthened RC beams ................................................... 77

M. Januszkievicz, F. Pesavento, W. Grymin, D. Gawin
Modelling the strains induced by Delayed Ettringite Formation in cement-based materials ........................................ 79

M. Koniorczyk, D. Gawin
Modelling the frost-induced damage in fully saturated cement-based materials ...................................................... 81

E. Korol, J. Tejchman
FE analyses of a coupled energetic-statistical size effect in concrete beams under bending ......................................... 83

A. Kotarski, Z. Więckowski
Two-dimensional FE analysis of confined concrete column ......................................................................................... 85

J. Kozicki, J. Tejchman
Investigations of vortex and anti-vortex structures in sand during plane strain compression by DEM .............................. 87

M. Königsberger, B. Pichler, C. Hellmich
Micromechanics of hydrating cement pastes considering progressive C-S-H gel densification .................................... 89

I. Marczewska, J. Rojek, R. Kačianauskas
Investigation of micro-macro relationships of elastic parameters in the discrete element model of granular material ............. 91

S. Pietruszczak, E. Haghighat
Description of damage process in sedimentary rocks ................................................................................................. 93

J. Podgórski, J. Gontarz
Explanation of the mechanism of destruction of the cylindrical sample in the Brazilian test ....................................... 95

V. Sakharov
Dynamic behaviour of Zymne Monastery Cathedral on soil base with consideration of non-linear deformation of materials .... 97

L. Skarżyński, M. Nitka, J. Tejchman
Modelling of concrete fracture at aggregate level using FEM and DEM based on real microstructure ............................. 99

M. Wojciechowski
Numerical homogenisation of permeability coefficient for Darcy flow in porous media ........................................... 101
<table>
<thead>
<tr>
<th>Topic</th>
<th>Authors</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Notch type on material behaviour under monotonic tension</td>
<td>T. Szymczak, Z.L. Kowalewski, A. Brodecki</td>
<td>An influence of notch type on material behaviour under monotonic tension</td>
<td>149</td>
</tr>
<tr>
<td>Transformation-induced creep and relaxation of TiNi shape memory alloy</td>
<td>K. Takeda, R. Matsui, H. Tobushi, E. Pieczyska</td>
<td></td>
<td>151</td>
</tr>
<tr>
<td>Influence of nitrogen ion implantation on fatigue of a TiNi shape memory alloy tape</td>
<td>K. Takeda, R. Matsui, N. Levintant-Zayonts, S. Kucharski</td>
<td></td>
<td>153</td>
</tr>
<tr>
<td>Numerical and experimental analysis of a cool thermal storage unit</td>
<td>M.M. Zajęc, J. Karwacki, R. Kwidziński</td>
<td></td>
<td>155</td>
</tr>
<tr>
<td>Frequency-dependent temperature and strain evolutions of NiTi wire during cyclic stress-controlled martensitic transformation</td>
<td>L. Zheng, Y. He, Z. Moumni</td>
<td></td>
<td>157</td>
</tr>
<tr>
<td>Available numerical implementations of isogeometric analysis</td>
<td>Z. Kacprzyk, K. Ostapska-Łuczkowska</td>
<td></td>
<td>159</td>
</tr>
<tr>
<td>Comparison of IGA and FEM for the Poisson benchmark PDE</td>
<td>M. Łuczkowski, K. Ostapska-Łuczkowska, W. Cecot</td>
<td></td>
<td>161</td>
</tr>
<tr>
<td>A mesh-free particle model for simulation of trimming of aluminum alloy sheet</td>
<td>L. Bohdal, R. Patyk</td>
<td></td>
<td>165</td>
</tr>
<tr>
<td>Measurement aided computation of extensible cable deflections</td>
<td>W. Cecot, S. Milewski, J. Orkisz</td>
<td></td>
<td>167</td>
</tr>
<tr>
<td>Accuracy of Lattice Boltzmann Method in application to multiphase tribological flows</td>
<td>M. Dzikowski, J. Rokicki</td>
<td></td>
<td>169</td>
</tr>
<tr>
<td>A numerical scheme of shift-periodic boundary condition for LBM</td>
<td>A. Grucelski, J. Pozorski</td>
<td></td>
<td>171</td>
</tr>
<tr>
<td>Coupling of Finite Element Method and meshless finite difference method with nonconforming approximation orders</td>
<td>J. Jaśkowiec, S. Milewski</td>
<td></td>
<td>173</td>
</tr>
<tr>
<td>On the application of multipoint meshless method to the nonlinear analysis</td>
<td>I. Jaworska, J. Orkisz</td>
<td></td>
<td>175</td>
</tr>
<tr>
<td>Flow patters generated by a flapping airfoil</td>
<td>T. Kozłowski, H. Kudela</td>
<td></td>
<td>177</td>
</tr>
<tr>
<td>Collapse vortices and filamentary structures</td>
<td>H. Kudela</td>
<td></td>
<td>179</td>
</tr>
<tr>
<td>Vortex-in-cell method and parallel computations</td>
<td>H. Kudela, A. Kosior</td>
<td></td>
<td>181</td>
</tr>
<tr>
<td>The meshless procedure for the stream function-vorticity formulation of the Navier-Stokes equations</td>
<td>M. Mierzwiczak</td>
<td></td>
<td>183</td>
</tr>
<tr>
<td>Discrete Element Method in the influence study of faults of concrete specimens on uniaxial compression test</td>
<td>T. Nowicki</td>
<td></td>
<td>185</td>
</tr>
<tr>
<td>Application of the Okubo-Weiss parameter to dynamical resolution adjustment in the Smoothed Particle Hydrodynamics approach</td>
<td>M. Olejnik, K. Szewc, J. Pozorski</td>
<td></td>
<td>187</td>
</tr>
<tr>
<td>Modelling of transient heat transport in a two-layered crystalline solid films using the interval lattice Boltzmann method</td>
<td>A. Plasecka-Belkhayat, A. Korczak</td>
<td></td>
<td>189</td>
</tr>
<tr>
<td>Is the motion of a single SPH particle droplet/solid physically correct?</td>
<td>K. Szewc, K. Walczewska-Szwez, M. Olejnik</td>
<td></td>
<td>193</td>
</tr>
<tr>
<td>On the elaboration of a methodology to experimentally verify terminal ballistics models for small arms ammunition</td>
<td>D.E. Tria, R. Trębicki, J. Janiszewski</td>
<td></td>
<td>195</td>
</tr>
</tbody>
</table>
T. Wałęczak, G. Sypniewska-Kamińska
The method of fundamental solutions with optimization of source intensities approach ...................................................195

Z. Więckowski
Landslide modelling by the material point method ...........................................................197

MS08 organized by: T. Lewiński, B. Gambin
Mathematical Methods in Solid Mechanics, Biomechanics and Optimization – a Session in Honor of Prof. Joachim Telega in the 10th Anniversary of His Death

O. Adeleye, O. Fakinlede, J. Ajiboye, C. Adegbulugbe
Viscoelastic-viscoplastic material model for nonlinear deformation of dental resin composites ......................................199

P. Alawdin, J. Marcinowski
Analytical solution and numerical simulation of borehole ground heat exchangers for geothermal heat pump systems: ground influence zone ..................................................201

O. Ardatov, A. Maknickas, V. Alekna, R. Kačianauskas
Finite element stress analysis of lumbar vertebrae body during osteoporotic degradation .........................................203

O. Bar
Fast algorithm for flux around closely spaced non-overlapping disks ........................................................................205

W. Bielski, R. Wojnar
Laminar flow past the bottom with obstacles – from suspension to porous medium ....................................................207

R. Czapla
Simulations of random geometric objects on the plane and their applications ..........................................................209

S. Czarnecki, R. Czubački, T. Lewiński, P. Wawruch
The Free Material Design reduced to the Monge-Kantorovich problem .................................................................211

T. Gajewski, H. Steńak, K. Szajek, T. Łodygowski, M.G. Stanisic, G. Oszkinis
Numerical aspects of patient specific material calibration of human artery: case study using clinical data ................213

B. Gambin, E. Kruglenko, W. Secomski, P. Karwat
Temperature dependencies of ultrasound signals backscattered from an agar-oil soft-tissue mimicking material ........215

B. Gambin, E. Kruglenko, M. Byra, A. Nowicki, H. Piotrzkowska-Wróblewska, K. Dobruch-Sobczak
Changes in ultrasound echoes of a breast tissue in vivo after exposure to heat – a case study ........................................217

D. Gawel, P. Główka, M. Nowak
Digitally reconstructed radiograph procedure for modifying 3D model to meet the intraoperative vertebrae location .........219

S. Gluzman, V. Mityushev, W. Nawalaniec
Effective conductivity and critical properties of 2D composites ..................................................................................221

M. Gzik, W. Wolański, B. Gzik-Zroska, K. Joszko, M. Burkački, S. Suchoń
The impact of ergonomic factors influencing armoured vehicle crew safety .................................................................223

Safety analysis of passengers of public transport during frontal impact ........................................................................225

D. Kapanadze, G. Mishuris, E. Pesetskaya
Remarks on effective conductivity of nonlinear 2D doubly periodic composites ..............................................................227

D. Kim, R. Segev
Notes on the mechanics of the octopus’s arm .................................................................................................................229

E. Majchrzak, L. Turchan, G. Kałuża
Sensitivity analysis of temperature field in the heated tissue with respect to the dual-phase-lag model parameters ....231

V. Mityushev, W. Nawalaniec
Basic sums in description of random structures ........................................................................................................233

B. Mochnacki, E. Majchrzak
Numerical modeling of biological tissue freezing process using the dual-phase-lag equation ........................................235

K. Myślecki, J. Lewandowski
Modified Hu-Washizu principle as a general basis for FEM plasticity equations ..........................................................237

J. Nowak, M. Kaczmarek
Simulation of indentation test for lymphedematous tissue within poroelastic model ..................................................239

M. Pakuła
Identification of mechanisms of attenuation and dispersion of ultrasonic waves in cancellous bone – theory and experiment 241
I. Wardach-Święciecka, D. Kardaś
Numerical analysis of thermal decomposition of single solid fuel particle in a stream of hot fuel gases ............................................................. 293

Z. Wrzesiński
Changing the combustion area of powder grains in the engine of a two-chamber system ............................................................. 295

**MS10 organized by: K. Wilde, J. Chrościelęwski, M. Rucka, W. Witkowski**

**Mechanics in Engineering Problems**

A. Adamkowski, S. Henelik, W. Janicki, M. Lewandowski
Laboratory investigation of the influence of pipeline supports stiffness on water hammer and fluid-structure interaction ................................. 297

A.A. Sabouni-Zawadzka, W. Gilewski
Technical coefficients for continuum models of orthotropic tensegrity modules ............................................................. 299

A. Ambroziak
Experimental tests for the determination of mechanical properties of PVC foil ..................................................................................... 301

J. Badur, J. Chrościelęwski
On a four-time unification of Cosserat continua by the intrinsic approach ............................................................. 303

M. Banaszkiewicz
The creep behaviour of high-temperature rotating components with power-law constitutive models ........................................................................ 305

B. Blachowski, W. Gutkowski, P. Wiśniewski
Dynamic substructuring approach for human induced vibration of a suspension footbridge ............................................................. 307

A. Bogusławski, A. Tyliszczak, K. Wawrzak
Absolutely unstable round hot jet – a numerical study ......................................................................................................................... 309

J. Bukała, K. Damaziak, K. Kroszczyński, M. Krzeszowiec, J. Małąchowski, K. Sobczak
Analysis approach for a diffusor augmented small wind turbine rotor ................................................................................................. 311

J. Buśkiewicz
Geometric analysis of a 1-DOF, six-link feeder ................................................................................................................................. 313

W. Chajec, A. Dziubiński
Modal approach in the fluid-structure interaction ................................................................................................................................. 315

M. Chleciński, G. Jemielita
Free vibrations and buckling stability of micro-nonhomogeneous plate band resting on an elastic subsoil ............................................................. 317

Y. Chikahiro, I. Ario, J. Holnicki-Szule, P. Pawłowski, C. Graczykowski
Study on the optimization of the reinforced scissor type bridge ......................................................................................................................... 319

B. Chiliński, R. Pakowski
Analysis of bending and torsional vibrations of rotors with using perturbation methods ................................................................. 321

J. Chrościelęwski, W. Witkowski, B. Sobczyn, A. Sabik
First ply failure FEA of laminated shells undergoing large displacements – 6 parameter shell theory approach ................................................................. 323

B. Czado, B. Wrana
Method of prediction of load-settlement curve for a single pile ......................................................................................................................... 325

R.T. Dalewski, R. Jóźwiak, O. Kobyliański, K. Rafał, J. Szumbarski
Design of a low power wind turbine adjusted to near-ground higher turbulence ......................................................................................................................... 327

J.M. Djoković, R.R. Nikolić, J. Bujnak
Influence of the weld geometry on the Stress Intensity Factor (SIF) of the cylindrical welded joint subjected to complex load ................................................................. 329

Ł. Doliński, M. Krawczuk, M. Palacz, A. Żak
Detection of damages in a riveted plate .......................................................................................................................................................... 331

Ł. Doliński, M. Krawczuk
Application of experimental modal analysis and wavelet transformation for damage localisation in a composite wind turbine blade ......................................................................................................................... 333

A. Dróżdż, W. Elsner, A. Kępiński
Investigation of turbulent boundary layers at moderate Reynolds number in the vicinity of separation ......................................................................................................................... 335

P. Felisiak, K. Sibilski, W. Wróblewski
Nonlinear model of spacecraft relative motion in an elliptical orbit ......................................................................................................................... 337

A. Fityka, A. Ryfa, L. Walencki, Z. Buliński, W. Adamszcz
Numerical and experimental study of the car aerodynamics ......................................................................................................................... 339

Ł. Flis
Static and dynamic accidental load analysis of Jet Hoods .......................................................................................................................................................... 341
<table>
<thead>
<tr>
<th>Author(s)</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>I. Maciejewski, T. Krzyżyński</td>
<td>Modelling of the vibration reduction system used for protection of working machine operators</td>
</tr>
<tr>
<td>A. Madaj, W. Siekierski</td>
<td>Identification of defect factors for a road bridge made of pre-stressed concrete on the basis of static strength analysis</td>
</tr>
<tr>
<td>A. Mariak, M. Miśkiewicz, B. Meronk, K. Wilde</td>
<td>Reference FEM model for SHM system of cable-stayed bridge in Rzeszów</td>
</tr>
<tr>
<td>B. Markiewicz, M. Kułpa, L. Ziemiański</td>
<td>Calculated and measured dynamic properties of the FRP composite beam</td>
</tr>
<tr>
<td>M. Maślań, M. Pazdanowski, M. Snela</td>
<td>Numerically based quantification of internal forces generated in steel sway frame structures with flexible end-plate joints, exposed to fire</td>
</tr>
<tr>
<td>Ł. Mazurkiewicz, J. Małachowski, P. Baranowski, K. Damaziak, W. Pytel, P. Mertuszka</td>
<td>Experimental study</td>
</tr>
<tr>
<td>M. Miśkiewicz, Ł. Pyrzowski, K. Wilde, J. Chróścielewski</td>
<td>Numerical analysis and in situ tests of Grot Rowecki bridge in Warsaw</td>
</tr>
<tr>
<td>M. Miśkiewicz, K. Daszkiewicz, T. Ferenc, W. Witkowski, J. Chróścielewski</td>
<td>Validation tests and numerical simulations of full scale composite sandwich segment</td>
</tr>
<tr>
<td>A. Mleczek, P. Kłosowski</td>
<td>Numerical analysis of the carpentry joints applied in traditional wooden structures</td>
</tr>
<tr>
<td>W. Mucha</td>
<td>Real-time hybrid simulation using materials testing machine and Finite Element Method</td>
</tr>
<tr>
<td>M. Nalepka, Z. Zembaty, S. Kokot</td>
<td>Experimental evaluation of wavelet based damage monitoring of a reinforced concrete frame</td>
</tr>
<tr>
<td>P. Nazarko, L. Ziemiański, S. Noga, T. Markowski</td>
<td>Comparative analysis of compound annular plates vibration on the basis of numerical and experimental studies</td>
</tr>
<tr>
<td>Ł. Mazurkiewicz, B. Sobczyk, W. Witkowski, J. Chróścielewski</td>
<td>Three-point bending test of sandwich beams supporting the GFRP footbridge design process – validation analysis</td>
</tr>
<tr>
<td>A. Robak, E. Błazik-Borowa, J. Bęc</td>
<td>Numerical analysis of scaffolding stands with defects</td>
</tr>
<tr>
<td>G. Rzyńska, A. Skrzat</td>
<td>Modeling of aluminum extrusion process based on Bodner-Partom model</td>
</tr>
</tbody>
</table>
K. Bolanowski  
Influence of temperature on the creep limit of microalloyed steel containing Nb, V and N .......................................................... 495

M. Bucior, L. Gałda, F. Stachowicz, W. Zielecki  
The effect of technological parameters of shot peening on surface roughness of 51CrV4 steel ........................................................... 497

K. Grochowska, K. Marynowski  
Dynamic behaviour of three layer composite cantilever beam with viscoelastic core .......................................................... 499

J. Kyziol, A. Okniński  
Metamorphoses of resonance curves in systems of coupled oscillators ................................................................. 501

N. Movchan, A. Movchan, M. Brun  
Structured waveguides: Floquet waves and polarisers in elongated systems .................................................. 503

Active cloaking of an inclusion at resonant frequencies for membrane and elastic flexural waves .............................. 505

M. Piotński, I. Pokorska  
Discontinuous Galerkin method for cracked solids with chemical compositions .......................................................... 507

B. Powalka, P. Pawelko, Z. Grządziel  
Regenerative chatter stability as a design criterion in the design of rope threading lathe ............................................. 511

L. Skee, G. Jelenić  
A multi-layer beam finite element for mixed-mode delamination in 2D beams ................................................................. 513

A. Skrzat, F. Stachowicz, I. Sevostianov  
Numerical and experimental prediction of the yield condition for porous materials .......................................................... 515

L. Witk, A. Bednarz, F. Stachowicz, I. Smirnov, N. Kazarinov  
Influence of crack size on resonant frequency of compressor blade .......................................................... 517

**MS12 organized by: A. Soldati, J.P. Minier, B. Geurts, J. Pozorski**

**Modelling and Simulating Disperse Two-Phase Flows**

C. Henry, J. Minier  
A stochastic approach for the deposition and resuspension of complex multilayered structures .................................................. 519

A. Innocenti, S. Chibbaro, M.V. Salvetti, C. Marchioli, A. Soldati  
A stochastic model for Lagrangian particle tracking in large-eddy simulation velocity fields ........................................ 521

S. Kornet, J. Badur  
Partial evaporation and total cut-off wet steam region on the shock wave ................................................................. 523

C. Marchioli, A. Soldati  
Turbulent breakage of ductile aggregates ................................................................. 525

C. Marchioli, A. Soldati  
On the rotation of rigid fibers in turbulent channel flow ................................................................. 527

P. Tiutiuński, D. Kardasi, I. Wardach-Święciecka  
CFD simulation of two-phase flow in the bearing chamber ................................................................. 529

**MS13 organized by: H. Sanecki, M. Mrzygłód**

**Modelling and Simulation in Land Vehicles and Aircrafts**

K. Dziewiecki, L. Prochowski, K. Zielenka  
Modelling and experimental investigation of the motion of a microbus passenger in the space between seat rows during a road accident .......... 531

S. Guzowski, M. Michnej  
Fretting wear simulation in model studies ................................................................. 533

M. Kalinowski  
Inverted joined-wing multidisciplinary optimization ................................................................. 535

M. Lis, A. Dziubiński, C. Galinski, T. Goetzendorf-Grabowski  
Dynamic stability analysis of the inverted joined wing scaled demonstrator ................................................................. 537

J. Magiera  
A comparative study of the performance of the 4-slice Transverse/Oblique Slicing method for analysis of 3D residual stress in prismatic bodies .......................... 539
M. Romanowicz
Numerical assessment of failure mechanisms due to transverse loading in unidirectional fiber-reinforced polymers .................................................. 589

N. Rylko
Fractal behavior of the heat flux on the boundary of random composites ................................................................. 591

M. Sitko, Ł. Madej, K. Muszka
Concurrent CAFE model of static recrystallization during multi-pass hot rolling .................................................. 593

J. Szypuder, Ł. Madej
Material model development for numerical simulation of the incremental forming process .................................................. 595

J. Wiącek, M. Molenda
Geometric and mechanical Representative Elementary Volume for polydisperse granular materials .................................................. 597

M. Wierszycki, K. Szajek, T. Łodygowski, M. Nowak
Numerical verification of two-scale approach for cancellous bone modelling .................................................. 599

T.G. Zieliński
Multiscale modelling of the acoustic waves in rigid porous and fibrous materials .................................................. 601

MS15 organized by: G. Mishuris, A. Linkov
Numerical Modelling in Hydraulic Fracturing and Related Problems

A. Dobroskok, A.M. Linkov, L. Rybarska-Rusinek
On simulation and interpretation of seismicity accompanying hydraulic fractures .................................................. 603

J.K. Grabski, J.A. Kołodziej
Generalized Newtonian fluid flow and heat transfer in an internally finned tube .................................................. 605

D. Jaworski, A.M. Linkov, L. Rybarska-Rusinek
Almost analytical evaluation of influence coefficients for ordinary and edge power-type boundary elements .................................................. 607

A.M. Linkov
Modified theory, universal asymptotic umbrella and efficient simulation of hydraulic fracturing .................................................. 609

G. Mishuris, M. Wróbel
Numerical simulation of hydraulic fracture: particle velocity based approach .................................................. 611

M. Perkowska, G. Mishuris, M. Wróbel
Numerical modeling of hydraulic fractures for non-Newtonian fluids .................................................. 613

E. Rejwer, D. Jaworski
On propagation of closely located hydraulic fractures .................................................. 615

J. Steller
Cavitation resistance of structural materials according to the fractional approach .................................................. 617

E. Tuliszka-Sznitko, K. Kieleczewski
Taylor-Couette flow with radial temperature gradient .................................................. 619

MS16 organized by: T. Lewiński, B. Bochenek
Optimization of Structural Topology

G. Bitzas, G.E. Stavroulakis
Design and topology optimization of an aluminium alloy wheel .................................................. 621

B. Bochenek, K. Tajs-Zielinska
Optimization of structural topology using unstructured Cellular Automata .................................................. 623

K. Bobotowski, T. Sokół
New method of generating Strut and Tie models using truss topology optimization .................................................. 625

G. Borsuk, B. Dobrowolski, B. Tomaszewska
Numerical study of slotted orifices shape influence on the downstream pressure distribution .................................................. 627

K. Brudło, M.S. Nowak, M. Morzyński, P. Bronny
Biomimetic optimization – differences and similarities in comparison to the SIMP method .................................................. 629

S. Czarnecki, R. Czubacki, P. Wawruch
Stress based version of isotropic material design in two dimensions .................................................. 631

S. Czarnecki, P. Wawruch
Selected problems of numerical analysis of Free Material Design .................................................. 633
S. Czarnecki, R. Czubańcki, T. Lewiński
Topological optimization of spatial continuum structures made of a non-homogeneous material of cubic symmetry ................................................................. 635

L. Falach, R. Segev
On the optimization of hyper-stress fields ........................................................................................................... 637

E. Idczak, T. Stręk
Optimization of auxetic structures using MMA algorithm .................................................................................. 639

J. Jackiewicz
Optimization of structures of modern materials using a new hybrid evolution strategy ........................................ 641

T. Kueczech
Application of manufacturing constraints method to structural optimization of AEC thin-walled structures ........ 643

R. Kutyłowski, M. Szwecławicz
Topological optimization as a tool for road pavement structure analysis .......................................................... 645

T. Łukasiak
HS(ro) – an isotropic material interpolation scheme based on Hashin-Shtrikman variational bounds ................ 647

M.W. Mrzygłód
A new procedure of solution search stabilization for evolutionary topology optimization ................................ 649

A. Myśliński, M. Wróblewski
Structural optimization of contact problems using piecewise constant level set method .................................... 651

M.S. Nowak, H. Hausa, R. Roszak, M. Morzyński, K. Brudło
Biomimetic optimisation – new approach to aircraft structural design ............................................................ 653

M.J. Pazdanowski
On the decreasing of the optimization problem size ......................................................................................... 655

G.I.N. Rozvany, T. Sokół, V. Pomezanowski, Z. Gaspar
Extension of Michell’s classical (1904) truss topology optimization theory to multiple load conditions, stress and displacement constraints, space (3D) trusses, probabilistic design and discontinuous support conditions ............................................. 657

O. Savchenko
Optimization of dynamic characteristics of composite shells by using genetic algorithms ......................... 659

T. Sokół, G.I.N. Rozvany
A new adaptive ground structure method for multi-load spatial Michell structures ........................................ 661

V.E. Volkova
Phase trajectories of non-linear noised dynamic system ................................................................................. 663

MS17 organized by: B. Skoczeń, H. Altenbach, D. Weichert
Physics Based Modelling in Solid Mechanics

I. Berinskii, H. Altenbach
Dependence of the elastic properties of two-dimensional crystals on their curvature ......................................... 665

J. Bielski, B. Skoczeń
Modified constitutive model of discontinuous plastic flow in intermetallic composites ........................................ 667

G. Bolzon, P. Pandi
The influence of imperfect interfaces on the overall mechanical response of metal-matrix composites ................. 669

W. Egner, S. Mruziński, H. Egner, P. Sulich
Effect of temperature rate in modelling non-isothermal fatigue of steel .......................................................... 671

L. Fraś, R.B. Pęcherski
Viscoplastcity of magnetorheological materials – theoretical description and experimental investigations ........... 673

M. Gačeša, G. Jelenić
Objectivity of strain measures in the fixed-pole approach ............................................................................... 675

K. Kowalczyk-Gajewska, K. Frydrych, M. Maj, L. Urbański
Micromechanical modelling of magnesium alloy and its experimental verfication ........................................... 677

D. Kukla, Z.L. Kowalewski
Influence of aluminum layer thickness on the fatigue properties of super-nickel alloy ........................................ 679

D. Peck, M. Mishuris, M. Wróbel, Y. Petrov
An improved estimate for threshold fracture energy in solid particle erosion ................................................... 681
H. Petryk, M. Kursa
The energy approach to rate-independent plasticity of metal single crystals ................................................................. 683

U. Radvilaite, R. Kačianauskas, D. Rusakevičius
Application of spherical harmonics to symmetric non-spherical particles description .......................................................... 685

M. Ryś, H. Egner
A unified theory of elastic-plastic-damage material with plastic strain induced phase transformation ............................. 687

K. Santaoja
Thermodynamics of a material model showing creep and damage ............................................................................... 689

J. Tabin, B. Skoczeń
Thermal and dissipative effect accompanying discontinuous plastic flow ............................................................... 691

A. Ustrzycka, B. Skoczeń
Kinetics of evolution of radiation induced damage ..................................................................................................................... 693

W. Waszkowiak, A. Żak, M. Krawczuk
Modelling of periodic structures by spectral finite elements ........................................................................................................ 695

MS18 organized by: M. Cieszko, J. Kubik, M. Kaczmarek
Porous Materials – Theory, Numerical Simulations and Experiments

M. Chuda-Kowalska, M. Malendowski
Sensitivity analysis of behaviour of sandwich plate with PU foam core with respect to boundary conditions and material model 697

M. Cieszko, Z. Szczepański, M. Kempński, P. Gadzała, M. Burzyński
Application of Micro Computed Tomography and Mercury Porosimetry to determination of internal structure of aerated concrete 699

M. Cieszko, T. Bednarek, T. Czerwiński
Stationary flow of non-wetting liquid through layer of unsaturated porous material ....................................................... 701

M. Cieszko, M. Kempński
Application of capillary and random chain models in mercury intrusion porosimetry ................................................... 703

M. Cieszko
Macroscopic description of capillary transport of liquid and gas in unsaturated porous materials ................................ 705

Experimental verification of the relationships between Young’s modulus and bone density using Digital Image Correlation 707

L. Fańczewski, T. Łodygowski, T. Jankowiak
Numerical modelling of aluminium foam based on quasi-static compression test ................................................................. 709

J.K. Grabski, M. Mierzwiczak
Creeping flow of a power-law fluid through a fibrous porous media .............................................................................. 711

K. Kazimierska-Drobny, M. Kaczmarek
Chemo-mechanical and thermal behaviour of PVA hydrogels .................................................................................. 713

M. Pakuła, R. Drelich, M. Kaczmarek, J. Kubik
Studies of ultrasonic waves in water or air saturated high porosity materials .......................................................... 715

R. Studziński, Z. Pozorski, M. Chuda-Kowalska
Experimental and numerical analysis of sandwich panels with composite core .............................................................. 717

T. Wegner, D. Kurpisz
The energy criteria of plastic flow for aluminum foam in complex load state ................................................................. 719

MS19 organized by: M.M. Kamiński, J. Górski
Probabilistic Methods in Mechanics

E. Böhm, M. Kurek, T. Łagoda
Fatigue life assessment with the use of exponential and power law functions for variable amplitude loading ............................. 721

A. Dudzik, U. Radon
The reliability assessment of a steel industrial hall ........................................................................................................ 723

J. Gajewski, T. Mikulski
Structural sensitivity analysis of telecommunication tower .................................................................................... 725

M. Hammoutene, B. Tillouine, B. Benahmed
Numerical investigation of the effects of damping uncertainties on Algerian seismic code spectra by Monte Carlo simulation 727
C. Tran
Parallel computing using Multi Processor System-on-Chip (MPSoC) for structural damage detection in real time ............................................. 777

MS21 organized by: M. Kuczma, J. Schröder, G.E. Stavroulakis, G. Szefer
Smart Material Systems and Structures

S. Abu-Salih
An analytical study of electromechanical buckling of micro spherical thin film bonded to a spherical compliant substrate ............................................. 79

T. Bartel, B. Kiefer, K. Buckmann, A. Menzel
Application of quasiconvex analysis: enhanced micromechanical modelling of martensitic phase transformations and numerical implementation . . 781

A. Denisiwicze, M. Kuczma
Two-scale elastic-plastic model of RPC in the plane stress state ................................................................. 783

M.W. Dobry
Energy efficiency of vibroisolation with constant reaction force (VCRF) ....................................................................................... 785

D.K. Dusztakar, A. Menzel, B. Svendsen
Numerical modelling of the rate-dependent polarisation switching in ferroelectric materials based on a sequential laminate approach .......... 787

K. Kęcik, A. Mitura
Influence of active elements on the pendulum's rotational motion for energy harvesting .......................................................... 789

A. Denisiwicze, T. Bartel, A. Menzel
Application of quasiconvex analysis: enhanced micromechanical modelling of martensitic phase transformations and numerical implementation . . 781

A. Denisiewicz, M. Kuczma
Two-scale elastic-plastic model of RPC in the plane stress state ................................................................. 783

M. Kureš
Subgroups of jet groups and material symmetries .............................................................................................. 793

W. Kurnik, A. Perek, P.M. Przybyłowicz
Double-source flutter in a discrete-continuous rotor/bearing system with magnetic fluid ...................................................... 795

M. Lasecka-Plura, R. Lewandowski
Frequency response function of structures with viscoelastic dampers and its design sensitivity analysis ............................................................. 797

J. Lengiewicz, M. Kursa, P. Holobut
Actuation by reconfiguration-modular active structures to create Programmable Matter ............................................................. 799

A. Olszewska, Z. Pawlak
Influence of geometrical and physical irregularities on dynamic characteristics of a passively damped structure ............................................................. 801

R. Ostwald, T. Bartel, A. Menzel
A framework for the simulation of phase-transforming elasto-plastic SMA and TRIP steel polycrystals ............................................................. 803

D. Pilsarski, C.I. Bajer, B. Dyniewicz
Semi-active stabilization of smart structures subjected to impact excitation ............................................................. 805

J. Przybyłski
Analytical modelling of a piezoelectric displacement amplifier with two pairs of flexure hinges ............................................................. 807

J. Przybyłski
Analytical modelling of a piezoelectric displacement amplifier with two pairs of flexure hinges ............................................................. 807

W. Rączka, J. Konieczny, M. Sibielski, J. Kowal
Frequency domain model of active vibration absorber based on SMA spring ............................................................................. 809

J. Schröder, M. Labusch, M. Keip
Multiscale homogenization of magneto-electric composites: how the ferroelectric polarization affects the product properties ............................................................. 811

T. Szmidt, C.I. Bajer
Finite displacement dynamic model of twin beams with controllable damper ............................................................. 813

K. Tuma, S. Stupkiewicz, H. Petryk
Phase-field modelling of twinning and martensitic transformation at finite strain ............................................................. 815

A. Weremczuk, J. Warmiński, R. Rusinek
The influence of external excitation on the dynamics of milling process ............................................................................. 817

J. Włodzisiewski
Mechanism of bi-direction laser bending for micro systems ............................................................................. 819

MS22 organized by: T. Burczyński, L. Ziemiański
Soft Methods and Inverse Analysis in Mechanics of Structures and Materials
– a Session in Honor of Prof. Zenon Waszczyszyn in connection with His 80th Birthday and in the recognition of important scientific achievements in Mechanics

A. Długosz, P. Jarosz
Multiobjective optimization of electrothermal microactuators by means of Immune Game Theory MultiObjective Algorithm ............................................................. 821
<table>
<thead>
<tr>
<th>Authors</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>R. Drelich, B. Piwakowski, M. Kaczmarek</td>
<td>Influence of frequency range of surface waves on estimation of parameters of heterogeneous concrete using non-contact method</td>
<td>823</td>
</tr>
<tr>
<td>S. Duda, D. Gąsiorek, G. Gembalczyk, S. Kciuk, A. Mężyk</td>
<td>Design of fuzzy logic controller for a unloading system in mechatronic device for gait reeducation</td>
<td>825</td>
</tr>
<tr>
<td>J. Orkisz, M. Głownicki</td>
<td>On improving evolutionary algorithms applied to chosen problems of mechanics</td>
<td>827</td>
</tr>
<tr>
<td>K. Psiuś</td>
<td>Event-driven approximate reasoning</td>
<td>829</td>
</tr>
<tr>
<td>M. Śłoński</td>
<td>On-line identification of elastic parameters in composite laminates using Lamb waves</td>
<td>831</td>
</tr>
<tr>
<td>Z. Waszczyszyn</td>
<td>Identification of material parameters in thin elastic plates: basic problems of neural networks and Lamb waves applications</td>
<td>833</td>
</tr>
<tr>
<td>M. Jurek, L. Ziemiański</td>
<td>Damage detection and evaluation in GFRP strip based on elastic wave propagation and support vector machines classification</td>
<td>835</td>
</tr>
</tbody>
</table>

**MS23 organized by: C. S. Drapaca, S. Hartmann, J. Leszczyński, S. Sivalogathan, W. Sumelka**

<table>
<thead>
<tr>
<th>Authors</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>T. Błaszczyk</td>
<td>Derivation and numerical solution of fractional Euler-Bernoulli beam equation</td>
<td>837</td>
</tr>
<tr>
<td>J.E. Butzke, S. Bargmann</td>
<td>High-temperature deformation of polysynthetically twinned crystals of TiAl – numerical modeling of yield point</td>
<td>839</td>
</tr>
<tr>
<td>S. Hartmann, S. Rothe, M. Grafenhorst, P. Erbts, A. Düster</td>
<td>Theory and numerics of monolithic and partitioned thermo-mechanical coupling</td>
<td>841</td>
</tr>
<tr>
<td>J.A. Kołodziej, M. Mierzewiczak, J.K. Grabski</td>
<td>Computer simulation of the effective viscosity in Brinkman’s filtration equation using the Trefftz method</td>
<td>843</td>
</tr>
<tr>
<td>C. Liebold</td>
<td>Determination of elastic material parameters in higher-order continua based on size-dependent bending behavior of epoxy and SU-8</td>
<td>845</td>
</tr>
<tr>
<td>A.B. Malinowska</td>
<td>Generalized fractional calculus of variations and its applications</td>
<td>847</td>
</tr>
<tr>
<td>I.A. Morozov, L.A. Komar</td>
<td>Hyperelastic structural-mechanical model of filled rubber</td>
<td>851</td>
</tr>
<tr>
<td>Z. Nowak, M. Nowak, R.B. Pęcherski, M. Potoczek, R.E. Śliwa</td>
<td>Numerical simulations of mechanical properties of alumina foams based on computer tomography</td>
<td>853</td>
</tr>
<tr>
<td>J. Pamin, B. Wiśoło</td>
<td>Influence of heat conduction on instabilities in large strain thermoplasticity</td>
<td>855</td>
</tr>
<tr>
<td>R.B. Pęcherski, M. Nowak, Ł. Frąś</td>
<td>Numerical simulations of auxetic metallic foam fabrication process</td>
<td>857</td>
</tr>
<tr>
<td>K. Sharma</td>
<td>Volume fraction and finite-specimen size effects on a limited-permeable inclined crack in 2D magnetoelastic media using distributed dislocation method</td>
<td>859</td>
</tr>
<tr>
<td>P. Stapór</td>
<td>Modelling the solidification of a liquid flowing in a narrow pipe using XFEM</td>
<td>861</td>
</tr>
<tr>
<td>T. Stryęk, H. Jołek</td>
<td>Topology optimization of a two-phase core of a sandwich panel</td>
<td>863</td>
</tr>
<tr>
<td>W. Sumelka</td>
<td>Anisotropic fractional non-local model</td>
<td>865</td>
</tr>
</tbody>
</table>

**MS24 organized by: K. Magnuski, R. Mania, W. Pietraszkiewicz**

<table>
<thead>
<tr>
<th>Authors</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>A. Adamowicz</td>
<td>Thermo-mechanical stresses in a brake disc</td>
<td>867</td>
</tr>
<tr>
<td>Title</td>
<td>Page</td>
<td></td>
</tr>
<tr>
<td>----------------------------------------------------------------------</td>
<td>------</td>
<td></td>
</tr>
<tr>
<td>P. M. Lewiński, R. Rak</td>
<td>921</td>
<td></td>
</tr>
<tr>
<td>Soil-structure interaction of cylindrical water tanks with linearly varying wall thickness</td>
<td></td>
<td></td>
</tr>
<tr>
<td>S. Lychev, T. Lycheva</td>
<td>923</td>
<td></td>
</tr>
<tr>
<td>Theoretical and experimental study of thin-walled growing laminated structures</td>
<td></td>
<td></td>
</tr>
<tr>
<td>E. Magnucka-Blandzi, Z. Walczak</td>
<td>925</td>
<td></td>
</tr>
<tr>
<td>Buckling and vibrations of seven-layer beams with lengthwise corrugated main core</td>
<td></td>
<td></td>
</tr>
<tr>
<td>E. Magnucka-Blandzi, M. Rodak, Z. Walczak</td>
<td>927</td>
<td></td>
</tr>
<tr>
<td>Buckling and vibrations of sandwich rectangular plates with trapezoidal core and three-layer faces</td>
<td></td>
<td></td>
</tr>
<tr>
<td>M. Malinowski</td>
<td>929</td>
<td></td>
</tr>
<tr>
<td>Post-buckling analysis of orthotropic circular cylindrical shell with inner corrugated layer</td>
<td></td>
<td></td>
</tr>
<tr>
<td>J. Mańkowski</td>
<td>931</td>
<td></td>
</tr>
<tr>
<td>Numerical simulations and experimental study of work riveted joints occurring in semi monocoque structures</td>
<td></td>
<td></td>
</tr>
<tr>
<td>J. Mańkowski</td>
<td>933</td>
<td></td>
</tr>
<tr>
<td>Numerical simulation of micro-slip occurring in riveted joints of semi monocoque structures</td>
<td></td>
<td></td>
</tr>
<tr>
<td>J. Marcinowski</td>
<td>935</td>
<td></td>
</tr>
<tr>
<td>The role of imperfections in nonlinear buckling analysis of a spherical shell roof</td>
<td></td>
<td></td>
</tr>
<tr>
<td>O. Mijušković, L. Tugić, B. Ščepanović</td>
<td>937</td>
<td></td>
</tr>
<tr>
<td>Analytical solution to buckling problems of plates with different boundary conditions under combination of patch load and bending</td>
<td></td>
<td></td>
</tr>
<tr>
<td>P. Ostrowski, B. Michalak</td>
<td>939</td>
<td></td>
</tr>
<tr>
<td>Tolerance modelling of stability of thin plates with a dense system of ribs</td>
<td></td>
<td></td>
</tr>
<tr>
<td>D. Pawlus</td>
<td>941</td>
<td></td>
</tr>
<tr>
<td>Dynamic response of an annular plate with a variable three-layered structure</td>
<td></td>
<td></td>
</tr>
<tr>
<td>A. Perelmutter, V. Yurchenko</td>
<td>943</td>
<td></td>
</tr>
<tr>
<td>Shear stresses in hybrid thin-walled section: development of detail numerical algorithm based on the graph theory</td>
<td></td>
<td></td>
</tr>
<tr>
<td>W. Pietraszkiewicz</td>
<td>945</td>
<td></td>
</tr>
<tr>
<td>On the resultant six-field linear theory of elastic shells</td>
<td></td>
<td></td>
</tr>
<tr>
<td>B. Sarkabiri, A. Jahan, M. J. Rezvani</td>
<td>947</td>
<td></td>
</tr>
<tr>
<td>Multi-objective crashworthiness optimization of thin-walled conical groove tubes filled with polyurethane foam</td>
<td></td>
<td></td>
</tr>
<tr>
<td>R. A. Sauer, T. X. Duong, K. K. Mandadapu, D. J. Steigmann</td>
<td>949</td>
<td></td>
</tr>
<tr>
<td>A computational formulation for liquid shells based on C1-continuous finite elements</td>
<td></td>
<td></td>
</tr>
<tr>
<td>S. Shahravi, M. J. Rezvani, A. Jahan</td>
<td>951</td>
<td></td>
</tr>
<tr>
<td>Optimization of foam-filled grooved circular tubes for energy absorption using response surface method</td>
<td></td>
<td></td>
</tr>
<tr>
<td>M. Taczała, R. Buczkowski, M. Kleiber</td>
<td>953</td>
<td></td>
</tr>
<tr>
<td>Nonlinear analysis of functionally graded plates resting on elastic foundation using the higher order plate theory</td>
<td></td>
<td></td>
</tr>
<tr>
<td>R. Tarczewski, M. Święciak</td>
<td>955</td>
<td></td>
</tr>
<tr>
<td>Topological optimization of formwork meshes for free-form surfaces</td>
<td></td>
<td></td>
</tr>
<tr>
<td>E. Tertel</td>
<td>957</td>
<td></td>
</tr>
<tr>
<td>Buckling of sandwich conical shells including the plasticity and unloading</td>
<td></td>
<td></td>
</tr>
<tr>
<td>B. Tomczyk</td>
<td>959</td>
<td></td>
</tr>
<tr>
<td>A new tolerance model of dynamic problems for thin biperiodic cylindrical shells</td>
<td></td>
<td></td>
</tr>
<tr>
<td>A. Tylikowski</td>
<td>961</td>
<td></td>
</tr>
<tr>
<td>Stability of hybrid rotating shaft with imperfect boundary conditions</td>
<td></td>
<td></td>
</tr>
<tr>
<td>N. Vaysfel’d, G. Popov</td>
<td>963</td>
<td></td>
</tr>
<tr>
<td>Bending of a rectangular thick plate with respect to its proper weight</td>
<td></td>
<td></td>
</tr>
<tr>
<td>L. Wittenbeck, K. Magnuski</td>
<td>965</td>
<td></td>
</tr>
<tr>
<td>Elastic buckling of corrugated plates</td>
<td></td>
<td></td>
</tr>
<tr>
<td>L. Wittenbeck, P. Jasion</td>
<td>967</td>
<td></td>
</tr>
<tr>
<td>Buckling and vibrations of seven-layer beam with lengthwise corrugated main core – numerical study</td>
<td></td>
<td></td>
</tr>
<tr>
<td>P. Yukhymets, A. Shekero, G. Zecheru, A. Dumitrescu</td>
<td>969</td>
<td></td>
</tr>
<tr>
<td>Strength of a damaged T-joint under Low-Cycle Loading</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ł. Żmuda-Trzebiatowski, P. Iwicki, M. Krajewski</td>
<td>971</td>
<td></td>
</tr>
<tr>
<td>Investigation of stability and limit load of a truss overhead opened bridge</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Authors</td>
<td>Title</td>
<td></td>
</tr>
<tr>
<td>---------</td>
<td>-------</td>
<td></td>
</tr>
<tr>
<td>V. Bagdasaryan</td>
<td>Thermal stresses in elastic periodic laminates</td>
<td></td>
</tr>
<tr>
<td>Ł. Domagalski, J. Jędrysiak</td>
<td>Nonlinear dynamic response of periodically inhomogeneous Rayleigh beams</td>
<td></td>
</tr>
<tr>
<td>R. Idzikowski, K. Misiurek, P. Śniady</td>
<td>Dynamic response of sandwich beam with periodic core due to fuzzy stochastic moving load</td>
<td></td>
</tr>
<tr>
<td>J. Jędrysiak</td>
<td>Tolerance modelling of vibrations of visco-elastic thin periodic plates with moderately large deflections</td>
<td></td>
</tr>
<tr>
<td>J. Jędrysiak, E. Pazera</td>
<td>Tolerance modelling of thermoelastic phenomena in functionally graded laminates</td>
<td></td>
</tr>
<tr>
<td>K. Jeleniewicz, W. Nagórko</td>
<td>Free vibrations of plates reinforced by rods – the homogenization with micro-local parameters</td>
<td></td>
</tr>
<tr>
<td>D. Kula, A. Radzikowska, E. Wierzbicki</td>
<td>Impact of tolerance averaging of heat transfer equation into exact description of a boundary effect phenomenon</td>
<td></td>
</tr>
<tr>
<td>J. Marczak, J. Jędrysiak</td>
<td>Tolerance modelling of vibrations in three-layered periodic structures</td>
<td></td>
</tr>
<tr>
<td>B. Tomczyk</td>
<td>A new combined model of dynamic problems for thin uniperiodic cylindrical shells</td>
<td></td>
</tr>
<tr>
<td>M. Wągrowska, O. Szlachetka</td>
<td>Tolerance modelling of elastic-nonelastic multilayered two-component composites</td>
<td></td>
</tr>
<tr>
<td>M. Wągrowska, O. Szlachetka</td>
<td>Heat conduction in biperiodic rigid composites</td>
<td></td>
</tr>
</tbody>
</table>
Plenary lectures
Interactions between mechanical systems and continuum mechanical models in the framework of biomechanics and vehicle dynamics

Jorge Ambrósio
LAETA, IDMEC, Instituto Superior Técnico, University of Lisbon
Av. Rovisco Pais 1, 1049-001 Lisbon, Portugal
e-mail: jorge.ambrusio@tecnico.ulisboa.pt

Abstract

Multibody dynamics approaches provide some of the most general and efficient computational dynamics methodologies to model complex systems in which the relative large overall motion of the components play a major role. Initially addressing only systems made of rigid bodies with relative motion between the components described by perfect kinematic constraints the multibody systems now include the description of the deformation of the components and allow for the joints to be described by contact pairs with local deformations and tribological effects. Here, the traditional construction of multibody systems is first described being alternative formulations for kinematic joints, by perfect kinematic pairs or by contact joints, presented. The possibility for the deformation of the system components is also included in the multibody formulation by using the finite element method to discretize particular components that exhibit deformations that influence the overall performance. The modelling of complex systems by multibody dynamics and finite element methods is further expanded by developing a co-simulation procedure that enables sub-systems to be modelled and analysed using different methods and codes while maintaining the synchronism of their forward time integration. The methods overviewed are applied first to biomechanical models for the human upper and lower limbs, including their detailed musculoskeletal systems, in order to show not only the constructive elements of a traditional multibody model but also the importance of considering imperfect and contact joints instead of the classical kinematic relations. The demonstrative examples are further pursued in the framework of vehicle dynamics in which not only the modelling aspects of realistic mechanical joints are of importance for the system performance but also in which the deformation of the system components play a role. Generalized deformations of sub-systems of the complex multibody system are used in the framework of a satellite deployment for which some of the structural components are made of composite materials and include piezo-electric sensors and actuators for their active control. The use of co-simulation approaches for the simulation of the interaction between structural, or fluid, systems with multibody systems is finally demonstrated by the application to the study of the interaction between pantograph and catenaries for high-speed railway vehicles.

Keywords: multibody dynamics, kinematic constraints, contact joints, contact mechanics, flexible bodies, co-simulation

1. Introduction

Multibody dynamic formulations are the basis for the most efficient computational techniques that deal with large overall motion. Formulations based on nonlinear finite element methods provide powerful and versatile procedures to describe the flexibility of the system components. It is no surprise that many of the most recent formulations on flexible multibody dynamics and on finite element methods with large rotations share common features that are of fundamental importance to the design requirements of advanced mechanical and structural systems and to the real-time simulation of complex systems [1].

Figure 1: Generic representation of a multibody system

The framework for a multibody system, shown in Fig. 1, describes the general motion of rigid bodies with the Newton-Euler equations of motion. The kinematic relations between the different bodies of the system are included by using Lagrange multipliers [2]. For components that experience deformations which influence their dynamic performance the assumption of rigidity is not used being their flexibility described with respect to their local reference frames using the finite element method [3,4]. The remaining ingredients for the application of multibody dynamics to realistic complex systems involve the description of the contact between system components or with other systems via the force vector in the equations of motion.

 Currently, multipurpose software packages exploit the ease of use of the computational resources available to create virtual prototyping environments [5, 6]. These packages share common geometric modelling features with more or less intuitive interfaces to build multibody or finite element models. Applications to vehicles, deployable structures, space satellites, machines or robot manipulators, which undergo large rigid body motion and material or geometric nonlinear deformations are examples of the application of these approaches.

2. Rigid multibody systems

In a wide number of practical applications the flexibility of the system components does not play a role being enough to represent them as rigid bodies. Vehicle dynamics, biomechanics of human or animal motion or machine dynamics provide extensive examples of this type of systems.
2.1. Equations of motion and kinematic restrictions

The equilibrium equations for a multibody system made of rigid bodies is generally represented in the form of [2]

\[
\begin{bmatrix}
M & \Phi_q
\end{bmatrix}
\begin{bmatrix}
\dot{q}_s
\end{bmatrix}
=\
\begin{bmatrix}
g
\end{bmatrix}
\begin{bmatrix}
\Phi_q
0
\end{bmatrix}
\lambda
\gamma
\]

(1)

in which \( M \) is the system mass matrix, \( \dot{q}_s \) is the vector that contains the state accelerations, \( g \) is the generalized force vector, which contains all external forces and moments, except for the vector of the constraint reaction forces described by \( g^c = -\Phi_q^c \lambda \) and included in left-hand side of the equations. The set of unknown Lagrange multipliers are associated to the intensity of the joint reaction forces. It should be noted the form of Eqn. (1) implies the use of Cartesian [2] or of Natural [7] coordinates to represent the position and orientation of the body fixed coordinate frame associated to each rigid body.

The kinematic restrictions can be viewed as algebraic relations between the rigid body coordinates, as implied in the illustration of Fig. 2(a) or as contact joints between two bodies, as for the clearance joint depicted in Fig. 2(b).

Any joint that is described as a perfect kinematic joint is included in the equations of motion as a kinematic constraint via the term \( g^c = -\Phi_q^c \lambda \). However, in many problems of practical importance the mechanical, or biomechanical, joints are not kinematically perfect exhibiting clearances, and eventually local deformable elements such as bushings. In this case clearance contact joints are used being their influence on the equations of motion done via the force vector \( g \) and not by adding the term with the Lagrange multipliers [8].

The dynamics of many practical systems in the areas of vehicle dynamics, machines, biomechanics or space vehicles, can be properly addressed with the equations of motion in Eqn. (1) being the large load of the modelling efforts put in the description of the interaction of the system components and external or internal objects. The rail-wheel contact, in railway dynamics, the road-tire contact mechanics in automotive dynamics or the musculoskeletal system actuation, in biomechanics of motion, are examples of cases in which proper constitutive relations that describe the interaction phenomena are required.

2.2. Biomechanics of the shoulder and shoulder prosthesis

The dynamics of human and animal motion generally use multibody models of musculoskeletal systems and biological mechanisms, in which the anatomical segments are represented by rigid bodies and the muscles represented either by kinematic constraints or by actuators. The anatomical joints are generally represented by mechanical joints or, in particular applications, by contact joints. Practical applications focus the identification of the muscle and joint reaction forces developed in the human body to perform a given task or the design of prostheses that ensure not only a suitable biomechanical functionality but also a long term stability and durability.

A multibody model for the shoulder in which the anatomical joints are described by perfect mechanical joints and the muscles act along straight and curved segments is proposed in Ref [9] being some of its modelling features highlighted in Fig. 3. The biomechanical problem solved with this model consists on finding the internal muscle and joint reaction forces for prescribed motions acquired in a biomechanical laboratory.

The biomechanical model presented in Fig. 3 is further modified to include a reverse shoulder prosthesis, shown in Fig. 4(a) and identifying the muscle forces as well as the loads on the prosthesis ball and cup. The glenohumeral joint with the prosthesis is described by clearance ball joint, seen in Fig. 4(b), in which the clearance and the contacting surface material properties influence the shoulder dynamics [10]. Note that due to the techniques used in the surgery part of the deltoid muscle segments are deactivated leading to a difference on the musculoskeletal model with respect to that of the natural shoulder.

The model is further used to find the variation of the shoulder joint reaction forces, i.e., glenohumeral joint reactions, as a function of the different prosthesis geometries. Fig. 5 presents the variation of the glenohumeral reaction forces as function of the arm elevation for an unloaded abduction motion in the coronal plane. In the process of the application of the reverse prosthesis shoulder model issues such as the detection of impingement between the scapula and the humerus, the modification of the muscle insertion points or the deactivation of some muscle bundles are addressed, thus providing the surgeon with a powerful design tool to support the surgery procedure adopted.
2.3. Railway vehicle dynamics

In vehicle dynamics the studies for vehicle stability and manoeuvrability require realistic models for the kinematic joints, without which fundamental interaction phenomena cannot be identified. The study of the virtual approval for operation of a railway vehicle in a mountain track provides an application example in which the detailed modelling of the vehicle with its suspension systems, shown in Fig 4, and of the interaction with the track is of fundamental importance [11].

![Figure 6: Multibody railway vehicle: (a) vehicle on its tracks; (b) model for one of the boggies](image)

The industry specifies a series of limit values on the forces that the vehicle can apply on the rails, on the forces that the vehicle bogies or wheelsets can be subjected to, on the accelerations of the bogies, wheelsets or the carbody and on the dynamic loading that the occupants are subjected to. The evaluation of the criteria associated with homologation and acceptance lead to values that must be contained within the thresholds prescribed by the norm [12, 13].

The quality of the models and their ability to predict a realistic behaviour for the rail-wheel contact. In particular, the modelling decisions on the description of the kinematic joints of the bogies, depicted in Fig. 7, influence the vehicle ability to be inserted in the track and to keep the wheels in permanent contact. The analysis show that models in which the realistic joint clearances are not considered lead to premature derailment [11].

![Figure 7: Typical modelling of boggies joints: (a) restrictions by force elements; (b) clearance cylindrical joint](image)

The wheel to rail contact is supposed to occur most commonly between the wheel tread, represented in Fig. 8 by the red ball, and the rail and not between the flange and rail, whose closest points for potential contact are represented by two blue balls in Fig. 8. Not only the correct rolling contact model used for the wheel-rail contact representation is of crucial importance for the evaluation of the interaction forces, but also the relative kinematics between system components needs to be correctly described to allow for a correct wheel to rail insertion.

![Figure 8: Typical modelling of boggies joints: (a) restrictions by force elements; (b) clearance cylindrical joint](image)

3. Flexible multibody systems

The flexibility of the multibody system components plays an important role in the system dynamics response in light weight systems or in classical mechanical systems experiencing particular loading scenarios [3]. Although alternative procedures have been proposed, such as plastic hinges [14], finite segments [15] or meshless methods [16], multibody system that include flexible bodies, as that depicted in Fig. 5, include their flexibility described by using the finite element method.

3.1. Flexible body equations of motion

In the applications described presented here, the finite elements are described with respect to moving frames associated to the flexible bodies, represented by $\xi \eta \zeta$ in Fig. 5 [4]. The moving frame in this formulation plays the role of the inertial frame $XYZ$ in conventional finite element formulations.

![Figure 9: Flexible body including the local reference frame](image)

The most general equations of motion for a flexible body that can exhibit nonlinear deformations, due to geometric and material nonlinearities, are written as [4]

$$
\begin{bmatrix}
\mathbf{M}_r & \mathbf{M}_f \\
\mathbf{M}_f^T & \mathbf{M}_g
\end{bmatrix}
\begin{bmatrix}
\ddot{\mathbf{q}}_r \\
\dot{\mathbf{q}}_f
\end{bmatrix}
= \begin{bmatrix}
\mathbf{g}_r \\
\mathbf{g}_f
\end{bmatrix}
- \begin{bmatrix}
\mathbf{s}_r \\
\mathbf{s}_f
\end{bmatrix}
\begin{bmatrix}
\mathbf{0} \\
\mathbf{f} + \left( \mathbf{K}_\gamma + \mathbf{K}_{\text{ml}} \right) \Delta \mathbf{u} \end{bmatrix}$$

where $\mathbf{M}_r, \mathbf{q}_r$ are the rigid body equivalent equations of motion, as in Eqn.(1), and $\mathbf{M}_f, \dot{\mathbf{q}}_f = \mathbf{g}_f - \mathbf{s}_f \cdot \mathbf{f} + \left( \mathbf{K}_\gamma + \mathbf{K}_{\text{ml}} \right) \Delta \mathbf{u}$ are the standard nonlinear finite element equations of motion. In Eqn. (2) the nodal coordinates are denote by $\mathbf{u}^n$, where $\mathbf{u}^n$ denotes that the quantity $\mathbf{u}^n$ is expressed in the coordinates of the local frame, and $\mathbf{f}$ is the stress equivalent vector. The mass matrix sub-matrices $\mathbf{M}_r = \mathbf{M}_f^T$ effectively couple the large body motion with its deformations and vectors $\mathbf{s}_r$ and $\mathbf{s}_f$ are gyroscopic force terms associated to the description of the body flexibility with reference to a moving frame.

The flexible body equation of motion written in the form of Eqn.(2) does not provide a unique description for the displacement field requiring the application of reference conditions, i.e., the equivalent to the boundary conditions used in standard finite elements but with reference to the body fixed frame and not to the inertia frame. The boundary conditions are included in the flexible body equations of motion in the same way the kinematic constraints are included in the rigid multibody system equations, i.e., by using Lagrange multipliers. The equations of motion for a flexible body with the reference conditions included are written as [17]

$$
\begin{bmatrix}
\mathbf{M}_r & \mathbf{M}_f \\
\mathbf{M}_f^T & \mathbf{M}_g
\end{bmatrix}
\begin{bmatrix}
\ddot{\mathbf{q}}_r \\
\dot{\mathbf{q}}_f
\end{bmatrix}
= \begin{bmatrix}
\mathbf{g}_r - \mathbf{s}_r \\
\mathbf{g}_f - \mathbf{s}_f - \mathbf{f} - \left( \mathbf{K}_\gamma + \mathbf{K}_{\text{ml}} \right) \Delta \mathbf{u}
\end{bmatrix}$$

Any set of reference conditions must present at least six independent equations relating the nodal coordinates, i.e, the rigid body motion of the finite element mesh relative to the body reference frame must be removed. The boundary fixed
conditions, in which one or more nodes have fixed degrees-of-freedom, are probably the best known and used. However, the mean axis conditions [18], in which enforce the kinetic energy associated with the deformation, measured with respect to an observer stationed on the flexible body is minimized, and the principal axis conditions [19], which enforce that the body reference frame is located in the center of mass and that the axis orientation coincides with the principal inertia axis, are important alternatives that must be considered.

The size of the system described by Eqn. (3) and the nonlinearities involved makes the computational solution of a single body already expensive. When used in a multibody system, which may include a moderate or large number of flexible bodies, the use of Eqn. (3) may be computationally prohibitive for analysis with long time periods. That is why in most of the practical applications the deformations of the flexible bodies are elastic and small, the elastodynamic coordinates \( \mathbf{u}' \) can be reduced by using a substructuring technique, the mode component synthesis or the Craig-Bampton method [4, 20]. For conciseness, let the mode components synthesis be considered here. Let the nodal displacements of the flexible part of the body be described by a weighted sum of a set of assumed modes, which include the modes of vibration associated with the natural frequencies of the flexible body and some selected static modes of deformation

\[
\mathbf{u}' = \mathbf{X} \mathbf{w}
\]

By substituting the time derivatives of Eqn.(4) into Eqn. (3) and simplifying, the equation of motion for a flexible body experiencing linear elastic deformations with respect to its local reference frame is

\[
\begin{bmatrix}
\mathbf{M}_e & \mathbf{M}_b & 0 \\
\mathbf{X}'\mathbf{M}_b & \mathbf{X}'\mathbf{M}_b & \mathbf{X}'\mathbf{M}_b \\
0 & \mathbf{X}'\mathbf{M}_b & 0
\end{bmatrix}
\begin{bmatrix}
\dot{\mathbf{q}}_r \\
\dot{\mathbf{w}} \\
\dot{\lambda}^{(m)}
\end{bmatrix}
= \begin{bmatrix}
\mathbf{g}_s - \mathbf{s}_s \\
\mathbf{f}_s \\
\lambda^{(m)}
\end{bmatrix} - \mathbf{X} \left( \left( \mathbf{g}_s - \mathbf{s}_s \right) - \Delta \mathbf{w} \right)
\]

which represents 6 equations for the large gross motion plus \( nm \) equations due to the \( nm \) modes used in the modal composition.

3.2. Satellite deployment and structural active control

The use of the flexible multibody methodology is demonstrated in the modelling of the deployment of a satellite with structural components made of composite materials, shown in Fig. 10, and in its active control via piezoelectric sensors and actuators [11]. The objective is to find a reliable unfolding mechanism that can ensure not only that no interference exists during the process but also that the loading in all components remains within admissible limits.

![Figure 10: Satellite model with composite materials flexible bodies](image)

All components of the satellite, beams and panels, are made of composite materials being modelled here as independent flexible bodies. The unfolding mechanism is envisaged by implementing two actuators in prescribed joints as seen in Fig. 10. When the actuation laws of such actuators is developed with regards only to the kinematics of the different bodies, as if rigid bodies, a very large interference between the unfolding panels, shown in Fig. 11, is observed.

![Figure 11: Satellite model with composite materials flexible bodies](image)

Two types of corrective measures are tested: modification of the panel stiffness by optimization of the composite materials of the panels and beams that minimizes the maximum deformation energy; active control of the deformation of the panels by including two piezoelectric patches, one sensor and other actuator as depicted in Fig. 12.

![Figure 12: Satellite model with composite materials flexible bodies](image)

![Figure 13: Satellite model with composite materials flexible bodies](image)

3.3. Co-simulation of multibody and finite element models

Multibody dynamics provides an almost perfect framework for the description of the interaction between systems described by different equilibrium equations and/or analysed with different codes. The interaction between vehicles and infrastructure [21], as in the case depicted in Fig. 14(a) for the railway pantograph-catenary contact, or between fluid and structures, as in wind power generator [22], as in Fig. 14(b) exemplify cases in which each of the systems are modelled and simulated in different codes that must advance in time synchronously. The co-simulation between multibody and finite element dynamics codes is explored here to address this particular class of interaction between different systems.
The interaction of the pantograph and catenary is achieved through the contact of the pantograph registration strip on the catenary contact wire. The ability of collecting reliable data on the contact forces to allow not only monitoring the operating conditions of the overhead system but also to allow for the validation of numerical models is one of the important key issues of the pantograph/catenary dynamics. The catenary, shown in Fig. 15(a) is a stationary structure mostly made out of beams that exhibits important structural vibrations being those on the contact wire of particular importance. The pantograph, shown in Fig. 15(b) is mechanical system with a large range of motion.

The catenary systems models must be include the proper description of their deformations being the finite element method irreplaceable for the purpose. The length of each section of the catenary, as that represented in Fig. 16(a) are in excess of 1.5 km not being uncommon to require models with two or more sections. As for the pantograph multibody models, the mechanical components can be represented by rigid bodies, as shown in Fig. 16(b), as their deformations do not play a role in the interaction dynamics.

Being the catenary modelled by complex finite element models it is expected that its dynamics is solved by using suitable finite element codes. By the same token, the multibody pantograph models are handled by multibody dynamics codes. As the integration algorithms of these codes are not only different but also involve diverse time stepping strategies, their co-simulation is required [23]. The basic idea is that the finite element code supplies the multibody code with the contact forces between the two systems and receives, in return, the positions and velocities of the multibody system, as shown in Fig. 17.

Typical results of interest for the pantograph-catenary interaction analysis are shown in Fig. 18 for a scenario of a highspeed railway vehicle with multiple pantographs.
The variation of the contact force shows a relative periodicity compatible with the pantograph passage under droppers and steady arms but it does not provide much more useful information. The mean contact force, of 157N for an operation at 300 km/h, is required for homologation. The standard deviation of the contact force must be lower than 30% of the mean contact force, which is also verified. The uplift of the contact wire under the steady arm is well below the usual maximum of 10 cm allowed for this type of operations. Finally, it is observed that the droppers are almost unloaded when the pantographs run under them, as expected. Therefore, the results collected enable to decide for the acceptance of the compatibility of the pantograph with the catenary for the type of operation analysed here.

4. Conclusions

The methodologies for the analysis of multibody systems, presented in this work, show a versatility that allows for their use not only to handle complex systems of rigid bodies with restricted relative motions, but also to include their elastodynamics or to be used in co-simulation environments with methods for dynamic analysis typical to other areas. The decisions on the modelling aspects related to the description of the kinematic restrictions, achieved either by kinematic constraints or by contact forces that explore the clearance of the joints between the adjacent bodies are crucial to allow, or prevent, that particular type of analysis are performed. Also, the introduction of the elastodynamics of the different components of the system is naturally done in the framework by using a finite element description. Contrary to the traditional finite element methodologies, the flexible multibody dynamics represents correctly the coupling between the small body deformations and the gross overall motion. The different aspects of the use of multibody dynamics was finally demonstrated in the framework of applications in biomechanics, vehicle dynamics or spacecrafts.

References

Multiscale mechanics of dynamical metamaterials

Marc Geers\textsuperscript{1}, Varvara Kouznetsova\textsuperscript{2}, Ashwin Sridhar\textsuperscript{3}, Anastasiia Krushynska\textsuperscript{4}

\textsuperscript{1,2,3,4}Department of Mechanical Engineering, Eindhoven University of Technology
P.O. Box 513, 5600 MB Eindhoven, The Netherlands
e-mail: mg@tue.nl

Abstract

This contribution focuses on the computational multi-scale solution of wave propagation phenomena in dynamic metamaterials. Taking the Bloch-Floquet solution for the standard elastic case as a point of departure, an extended scheme is presented to solve for heterogeneous visco-elastic materials. The physically and geometrically nonlinear case is addressed through a transient computational homogenization scheme. In the particular case of an elastic heterogeneous microstructure, the homogenization scheme can be reduced to the computational analysis of a fluctuation-enriched extended continuum.

Keywords: Multi-scale, Computational Homogenization, Acoustic Metamaterials, Local Resonance, Wave Dispersion

1. Introduction

The mechanical behaviour of materials across the scales often inherits particular properties that are rooted in the intrinsic fine scale level. This is typically the case for problems where the micro-scale reveals special characteristics or pronounced inhomogeneities in the deformation micro-fluctuation fields. A particular class of such materials are acoustic metamaterials, designed to attenuate sound wave propagation for certain frequencies. The unique features of these metamaterials originate from the complex interaction of transient phenomena at the microscopic and macroscopic scales, with local resonance occurring within (one of the) micro-constituents resulting in effective band gaps at the macro-scale.

Band-gap materials, like phononic crystals (using Bragg scattering) and acoustic metamaterials (using local resonance), are typically used to control the propagation of elastic waves, where they reveal advanced properties such as negative effective mass density or negative effective elastic moduli. They typically exhibit frequency band gaps (i.e. ranges of frequencies), for which no propagating waves exist. This offers a great potential for many practical engineering applications.

2. Elastic and visco-elastic band-gap metamaterials

Band structures for solid metamaterials are often calculated under the assumption of linear elastic behaviour of the constituent components \[1\]. In practice, polymer-based visco-elastic materials are frequently used in these materials, which are in nature dissipative and time-dependent. The influence of the material dissipation on the wave dispersion has been studied for phononic crystals, but the methods used are quite limited when applied to acoustic metamaterials. New methods capable of dealing with a realistic viscoelastic response (e.g. generalized Maxwell) are therefore of great interest.

The propagating wave in a deformable continuum complies with the standard equations of motion, linear kinematic strain-displacement relation, and constitutive equations. In the case of a linear viscoelastic material, the constitutive equations can be expressed in terms of Duhamel integrals with time-dependent bulk/shear moduli. For time-harmonic waves, the Bloch-Floquet theorem is generally used to reduce the infinite periodic metamaterial domain to a representative unit cell. The Bloch periodic boundary conditions are applied at the unit cell boundary, whereby the resulting problem is non-linear in the time domain. However, the particular form of the viscoelastic constitutive relations permits a simplified description of all the equations in frequency domain by means of the classical elastic-viscoelastic principle \[2\]. The transition from the time domain to the frequency domain is achieved by means of the Laplace-Carson transform, which retains the form of the field equations by preserving the physical meaning of the involved coefficients. This allows to extend the existing methods for elastic metamaterials to the viscoelastic regime, whereby the field characteristics will depend on frequency. The considered unit cell of a metamaterial is discretized with a FE-scheme and band structures are calculated using the so-called \(k_c\)-formulation which has already been successfully applied to phononic crystals with visco-elastic components \[3\]. Real frequencies and complex-valued wavenumbers are used to describe the spatial attenuation of propagating waves. Compared to the literature, the proposed method works with any viscoelastic model and is numerically stable.

3. Nonlinear dynamic metamaterials

The methodology commonly used for elastic materials, and its extension to visco-elastic materials, cannot be applied for heterogeneous materials that present physical and geometrical non-linearities. Moreover, scale separation generally does not apply here, and the response of the material is dependent on the structure in which it is embedded. This class of metamaterials is therefore largely unexplored, since it requires a proper methodology to study the transient behaviour accompanying wave propagation.

3.1. Computational Homogenization

To address the nonlinear response of complex multi-phase solids within a given structure, the Computational Homogenization method is a powerful method \[4\]. This method is essentially based on the nested solution of two boundary value problems, one at each scale. Though computationally expensive, the proce-
dures developed allow to assess the macroscopic influence of microstructural parameters in a rather straightforward manner. The first-order technique is by now well-established and widely used in the scientific and engineering community. Several extended schemes have been proposed as well:

- Higher-order computational homogenization
- Continuous-discontinuous homogenization-localization, which aims to incorporate the transition from damage to fracture (via localization) in a multi-scale approach
- Thermomechanical computational homogenization
- Substructured thin sheets and shells
- Multi-scale interfaces or cohesive cracks
- Contact and friction problems

3.2. Computational homogenization for dynamical metamaterials

The solution in the nonlinear regime is based on a novel transient computational homogenization procedure addressing the evolution in space and in time of materials with a non-steady state microstructure [5], schematically depicted in figure 1.

The proposed transient scheme extends the classical concepts used in a first-order computational homogenization framework. The separation of scales hypothesis is relaxed, enabling solutions that go beyond the long wavelength approximation for the microstructural components. It is based on an enriched description of the micro-macro kinematics that allows large spatial fluctuations of the microscopic displacement field (relative to its macroscopic counterpart), originating from local resonance phenomena. An extended Hill-Mandel macro-homogeneity condition is proposed, which couples the coarse scale stress tensor and linear momentum to the solution of the underlying micro-scale problem. Accordingly, the balance of linear momentum is solved at both scales. This novel multi-scale solution method enables innovative designs of finite-size microstructures for locally resonant acoustic metamaterials.

4. Fluctuation-enriched extended continuum

In the case of a heterogeneous elastic microstructure, the homogenization method allows to eliminate the fine scale, by solving an extended macroscopic continuum, see figure 2.

![Figure 2: Homogenization towards a fluctuation-enriched extended continuum](image)

Using the Craig-Bampton reduction technique, internal (fine scale) degrees of freedom can be decomposed into their quasi-static and dynamic parts, and the homogenization method allows to define a (reduced) extra kinematic field \( \eta \), i.e. a dynamic fluctuation field, to be solved for through a coupled dynamic equation at the level of the representative volume element. This enrichment naturally accounts for the local resonance dispersion phenomena. The working principle of this method will be presented, along with an illustrative example.

References


Consistent plate theories – A matter still not settled?

Reinhold Kienzler¹, Patrick Schneider²

¹²Bremen Institute of Mechanical Engineering (bime), University of Bremen, Department of Production Engineering
Am Biologischen Garten 2, 28359 Bremen
e-mail: rkienzler@uni-bremen.de, pasch@uni-bremen.de

Abstract

Using the uniform-approximation technique in combination with the pseudo-reduction method, a hierarchy of consistent plate theories is derived. After the introduction of non-dimensional quantities, the strain-energy and the dual-energy densities appear as infinite power series in the plate parameter that describes the relative thinness of the structure. The associated Euler-Lagrange equations deliver a countably infinite set of PDEs, where each PDE is an infinite power series with respect to the plate parameter. It is shown that the untruncated set of PDEs is equivalent to the problem of the three-dimensional theory of elasticity. Furthermore, an a-priori error estimation is given for the truncated, finite and therefore tractable PDE system. The error of the $N$th-order two-dimensional theory decreases like the $(N + j)$th-power of the characteristic plate parameter, so that a considerable gain of accuracy could be expected for higher-order theories. The resulting equations of a consistent 2nd-order plate theory are used to assess and validate theories established in the literature.

Keywords: consistent plate theories, uniform-approximation technique, pseudo-reduction method, second-order theory

1. Introduction

Although it is generally accepted that the classical Kirchhoff-plate theory is a consistent first-order approximation of the linear three-dimensional theory of elasticity, a variety of higher-order theories exist that take shear deformations, cross-sectional warping and normal stresses in thickness direction into account. The variety of existing shear-correction factors (cf., e.g., [2]) may be regarded as an indication for the problem to be still unsettled.

In the talk we propose to use a theory based on the combination of the uniform-approximation technique [3] and the pseudo-reduction method [7]. In contrary to the direct approach (cf., e.g., [1]), it belongs to the class of theories, which use series expansions in thickness direction with respect to a proper basis to derive hierarchical sets of approximative theories from the classical equations of the linear three-dimensional theory of elasticity. After introducing non-dimensional quantities, the plate parameter $c^2 = h^2/12a^2$ evolves quite naturally, with $h$ and $a$ denoting the characteristic out-of-plane and in-plane measures, respectively. For plate theories, $c^2$ is usually assumed to be a small quantity, i.e., $c^2 \ll 1$. The infinite PDE system for the determination of the unknown coefficients of the series expansion, where each PDE appears as an infinite power series with respect to the plate parameter $c^2$, is exact, i.e., it is equivalent to the equations of the three-dimensional linear theory of elasticity.

The idea of the uniform approximation technique of an $N$th-order plate theory is to consider all terms multiplied by $c^{2m}$, $n \leq N$ and neglect terms multiplied by $c^{2m}$, $m > N$, i.e., terms that are of the order $O(c^{2N+1})$. We obtain a finite PDE system of $3(N + 1)$ equations for $3(N + 1)$ unknown displacement coefficients. During the reduction of these equations, i.e., elimination of unknowns, terms of the order $O(c^{2N+1})$, are also neglected (pseudo-reduction technique, for details cf. [7]). The procedure results in PDEs, which are derived with recourse to neither a-priori assumption nor shear-correction factors.

By invoking the strain-energy density as well as the dual-energy density, an a-priori error estimate is established for the uniformly truncated series expansion [6]. The error of the $N$th-order two-dimensional (plate) theory decreases like the $(N + 1)$th-power of the characteristic plate parameter $c$.

2. Second order theory

The derivation of the governing equations by using either monomic polynomials or scaled Legendre-polynomials has already been published [3, 4, 9]. In the following, we will concentrate on isotropic material behaviour, and we will use monomic polynomials as basis. In extension of these earlier investigations we introduce here energetic averages $\overline{\psi}$ and $\overline{\psi}_w$ of the transverse displacement $\psi$ and the slopes $\psi_w$ of the plate middle surface, respectively, as

$$c^2 \overline{\psi} = c^2 \psi + \frac{3}{10} v c^4 \Delta \psi + O(c^6),$$

$$c^2 \overline{\psi}_w = c^2 \psi_w - \frac{3}{10} (1-v) c^4 \Delta \psi + \frac{6}{5} \varepsilon_{\text{eul}} c^2 \psi_{\text{eul}} + O(c^6)$$

($\Delta$ is the two-dimensional Laplace operator, $\Delta(\cdot) = \partial_x^2 + \partial_y^2$ and $\varepsilon_{\text{eul}}$ is the completely screw-symmetric permutation tensor). The quantity $\psi$ is defined as $\psi = \psi_{\text{eul}} - \psi_{\text{rot}} \hat{z}$ and may be regarded as a measure of the transverse-shear deformation. This $\psi$ is next to $\overline{\psi}$ our of our two main variables which are governed (after pseudo reduction) by our two main PDEs

$$c^2 \left( \psi - \frac{6}{5} c^2 \Delta \psi \right) = 0 + O(c^6),$$

$$\varepsilon_{\text{eul}} \left( \Delta \psi_{\text{eul}} - \partial_x^2 \Delta \psi_{\text{eul}} \right) = c^2 \left( \psi_{\text{eul}} - \frac{6}{5} c^2 \Delta \psi_{\text{eul}} \right) + O(c^6),$$

cf. [5].

$K$ is the classical plate stiffness $K = E h^3/12(1-v^2)$ (Young’s modulus $E$, Poission’s ratio $v$) and $P$ is the transverse load applied through the upper and lower plate faces ($x_1 = \pm h/2a$).

The stress results are calculated as bending moments ($\delta_{\text{eul}}$, Kronecker’s tensor of unity)
\[
\begin{aligned}
\tilde{M}_{ab} &= K \left\{ \frac{1}{2} \left[ \frac{12}{5(1-v)} \left( 1-v \right) \tilde{w}_{,ab} + v \tilde{w}_{,a} \delta_{ab} \right] + \\
&+ \frac{3}{5} \left( 1-v \right) c^2 \left( \varepsilon_{x,ab} \psi_{,b} + \varepsilon_{y,ab} \psi_{,a} \right) \right\} \\
&+ \frac{6}{51-v} c^2 a^2 P \delta_{ab} + O(c^4)
\end{aligned}
\]  

(3a)

and transverse shear forces

\[
\tilde{Q}_b = K \left\{ \Delta \tilde{w}_{,b} + \frac{1}{2} \left( 1-v \right) \varepsilon_{,b} \psi_{,b} \right\} - \frac{6}{51-v} c^2 a^2 P \delta_{ab} + O(c^4).
\]

(3b)

Higher-order stress resultants are also involved and calculated but turn out to be expressible as linear combinations of the classical stress resultants.

The equilibrium equation deliver

\[
\frac{1}{a^2} \tilde{Q}_{a,b} = -P \Rightarrow (2a),
\]

\[
\frac{1}{a^2} \tilde{M}_{a,b} = \tilde{Q}_a \Rightarrow (2b).
\]

(4)

By the pseudo-reduction method, not all of the displacement-ansatz coefficients are determined. They may be chosen a posterior to fulfil the boundary conditions along the plate faces

\[
\sigma_{x1} \bigg|_{z=0} = 0 + O(c^4),
\]

\[
\sigma_{x1} \bigg|_{z=\pm a} = \pm \frac{1}{2} P + O(c^4),
\]

(5)

and the local equilibrium conditions

\[
\sigma_{a1,a} \sigma_{b1,b} = 0 + O(c^4),
\]

\[
\sigma_{a1,a} \sigma_{b1,b} = 0 + O(c^4),
\]

(6)

cf. [5].

Thus the proposed method leads a well-balanced and contradiction-free second-order plate theory without recourse to any a priori assumptions.

It may be mentioned that for anisotropic plates, i.e., plates of monoclinic materials, the equations for \( \tilde{w} \) and \( \psi \) are not decoupled anymore and involve 13 material constants [9].

3. Discussion

In the talk, the proposed second-order plate theory will be compared with other theories existing in the literature. We will assess and validate the theories of Reissner/Mindlin, Zhilin, Marguerre, Verkua, Ambartsumyan and Reddy, cf. [8].

References


Nanoscale challenges of fluid mechanics

Tomasz A. Kowalewski
Paweł Nakielski, Filippo Pierini, Krzysztof Zembrzycki, Sylwia Pawłowska

1,2,3,4,5Institute of Fundamental Technological Research, Polish Academy of Sciences
Adolfów Pawinskiego 5B, 02-106 Warszawa, Poland
e-mail: tkowale@ippt.pan.pl

Abstract

In this talk we would like to tackle general question of contemporary fluid dynamics, how far its assumption of a continuous, smooth medium remains useful when size and time scales start to approach molecular ones. The question is not trivial and seems to depend on several additional factors usually minored. For example, when full Navier-Stokes equations are replaced by their linear approximation we are losing basic characteristics of convective motion, and still we use such approach. Once our fluid becomes granular matter with its own internal properties, proper interpretation of flow interactions with other molecular structures probably needs deeper physics. But still we try to convert such problem to the classical macro/micro scale description. Hence a general question arises, how small does a fluid have to be before it is not a fluid anymore?

Keywords: microfluidics, nanofluids, Brownian motion, nanofilaments

1. Introduction

It is generally understood, that the main difference between macro- and nano-scale mechanics originates from rapidly increasing surface to volume ratio along with the decreasing of object size. A total surface of one nanometre particles filling volume of a cubic centimetre is 6000 square meters! Hence, nanoscience is mainly a science of surface forces and surface interactions. It applies particularly to fluids.

The field of microfluidics is characterized by the study and manipulation of fluids at the submillimetre length scale. The fluid phenomena that dominate liquids at this length scale are measurably different from those that dominate at the macroscale. For example, the relative effect of the force produced by gravity at microscale dimensions is greatly reduced compared to its dominance at the macroscale. Conversely, surface tension and capillary forces are more dominant at the microscale. These forces can be used for a variety of tasks, such as passively pumping fluids in microchannels, precisely patterning surfaces with user-defined substrates, filtering various analytes, and forming monodisperse droplets in multiphase fluid streams for a variety of applications [1].

It is interesting to note how many valuable hints on behaviour and explanation of micro-word indicates our mother nature, which evolutionary optimized the best nanoscale engineering system to control living organisms. For micro-fluidics surface tension is much more crucial, as this is the surface related force. This can be actually noticed observing bugs walking on the water surface. Their small size permits them to benefit from the surface tension. Another example is nanofibre web produced from polymeric solution exhibiting strength of the similar product produced in nature by spiders. The spider web is an ideal material in terms of endurance. It can stop flying bug and still remains untouched. In macro-scale adequate web would have to stop a whole airplane without damage, which obviously is still far from the current engineering capabilities.

By far the most difficult question which nanotechnology is facing is how to effectively produce tools and systems in nanoscale. And again the nature can help engineers to solve this problem. Any type of biological system is built from cells, which form more complex structure by organizing themselves. One of current research tasks is to study self-organization mechanism for efficient building of micro-systems. The macro-scale design techniques currently in use (top-down approach) such as lithography are relatively expensive and slow. Even, if prototypes of quantum computer are built of few atoms manipulated under electron microscope, such technique is completely inefficient for mass production. Separately manipulating millions of atoms to cover micro-meter surface would take several centuries. By self-organization this process can be achieved far more rapidly, in a matter of minutes or less. Hence, fluidic techniques based on patterned shapes of monolayers and capillary forces are used to assemble micro devices. In most cases the self-assembly requires that the components are mobile in a fluidic environment. Observing biology one obtains plethora of valuable hints how to proceed with such process [2].

Physics of nanoscale mechanics can be more or less completely described using quantum mechanics. The methods, called ab initio, are rigorous but limited by present-day computers to systems containing a few hundred atoms at most. To determine the properties of larger ensembles of atoms the Molecular Dynamics method is commonly used, enabling studies of billions of atoms with effective interatomic potentials. Still its practical applications are limited by time and space scales to first nanoseconds of the analysed phenomena and several nanometres in space. Hence, modelling fluid flow in micro- and nano-scales needs specific technique, which is based on assumption that fluid particle can be represented as a cluster of atoms. Effective clustering can be based on so called Voronoi tessellation, describing a special kind of decomposition of the flow domain [3]. Such coarse grained modelling is useful for general flow description, but needs predefined interactions if we approach molecular distances to interpret specific phenomenon. Hence, despite of technical and physical problems with direct, experimental analysis of nanoscale flows, it’s the only way to validate simplified assumptions which by definition have to be incorporated to theoretical models. Some experimental examples we try gain observing the mother nature, some of them has been recently proposed using

*Supported by NCN grant no. 2011/03/B/ST8/05481.
completely new for fluid mechanics techniques, like fluorescent microscopy, atomic force microscopy, and optical tweezers. Most of microfluidic problems concern multiphase flow, suspensions of micro and nanoparticles, cells or macromolecules (proteins, DNA etc). Understanding and properly interpreting fluid-particle interaction is crucial for interrogating such systems. One of the basic optical tools is based on Brownian motion [4]. The local and bulk mechanical properties of a complex fluid can be obtained by analysing thermal fluctuations of probe particles embedded within it.

Thermal fluctuations generated by molecules are not only noises, it has been demonstrated that such fluctuations are fundamental to the function of biological systems. Here, several noises, it has been demonstrated that such fluctuations are fundamental to the function of biological systems. Hence, its transport properties largely deviate from tube diameter below 1 nm, water forms a long single molecular structure of water. For constrained conditions, i.e. synchronized slippage of water molecules. It is interrelated with gases. For very small channels formed by carbon nanotubes (CNT) possible drag reduction is expected due to the synchronized slippage of water molecules. It is interrelated with molecular structure of water. For constrained conditions, i.e. tube diameter below 1 nm, water forms a long single molecular chain [9]. Hence, its transport properties largely deviate from continuous understanding of fluid flow.

2. To slip or not to slip

One of the primary questions, which appeared when fluid mechanists started to play with microfluidics, concerned the interactions between liquids and solid surfaces. From the physical point of view it seems obvious that molecules of liquid cannot be arrested at the solid surface, otherwise the local thermodynamic parameters of liquid should abruptly change. This problem is of fundamental physical interest and has practical consequences in rarefied gas flows. Recently it was rediscovered for small-scale systems, including transport phenomena in biological fluids [8].

The physics of hydrodynamic slip may have different origins. Purely molecular slip is clearly relevant in case of gases. For very small channels formed by carbon nanotubes (CNT) possible drag reduction is expected due to the synchronized slippage of water molecules. It is interrelated with molecular structure of water. For constrained conditions, i.e. tube diameter below 1 nm, water forms a long single molecular chain [9]. Hence, its transport properties largely deviate from continuous understanding of fluid flow.

Figure 1: Schematic definition of the slip velocity $U_s$ and the slip length $\lambda$ for the fluid flow over solid wall

For dense fluids several additional factors appear to play a more or less significant role. One of them involves wetting properties of the solid surface. Molecular scale roughness allows for creation and stabilization of nanobubbles. Such trapped on the wall nanobubbles may effectively work as a gaseous slip layer, responsible for super-hydrophobic surface properties [10].

Identification of the slip appears not a simple task, both numerically as experimentally. Molecular simulations performed up to now neither confirm nor exclude possible deviation from the classical “non-slip” condition formulated by Navier in 1823. Using continuum mechanics we have to combine a grid size that copes with a nanometer while covering enough space to also include a millimetre scale flow. Experimentally, our techniques for looking at the very small objects (e.g., atomic force microscopy) are slow and cannot cope with large areas very well. The attempts to look at both these scales simultaneously show how our intuition about the relationships between nanoscale objects and macroscopic objects can fail badly.

Classical microscopy used for nanoscale observation has resolution limited by the light wavelength of about 500 nm micrometres. Evaluating diffraction disks the measured position of particle coordinates in plane perpendicular to the optical axis can be improved by order of magnitude. However, resolution in depth, along optical axis, remains very low (tenths of micrometre), and is defined solely by focal depth of the microscope lens. Total Internal Reflection Microscopy (TIRF) helps to bypass some of these limitations offering possibility to locate objects position with resolution of about 20 nm. Laser light illuminating object undergoes total internal reflection at a interface between investigated medium (liquid) and the wall (glass), and part of the light penetrates into the medium parallel to the interface with an intensity that decays exponentially with the normal distance from the interface (Fig. 2). This evanescent wave illumination has been used extensively in the life sciences. Recently it was rediscovered in microfluidics for near wall flow measurements. The main advantage of the method is possibility to reduce the depth of focus of the acquisition system [11]. Hence, it became possible to obtain images of particles, which are in the direct vicinity of the wall. In our recent study of the Brownian motion of fluorescent particles observed close to the wall, the deviation of the particle diffusion rate has been interpreted as an evidence of the slip boundary conditions.
In the numerical analysis, for much smaller particles (24 nm) measured at 170 nm from the wall appears to be nearly 300 nm. Experiments (300 nm diameter) the evaluated slip length 3.

became partly understandable if we look at inherent factors measurements of slip length performed by particle tracking performed by Molecular Dynamics approach.

The outcome is in the range of uncertainty found in the literature. For relatively large nanoparticle used in the experiments (300 nm diameter) the evaluated slip length measured at 170 nm from the wall appears to be nearly 300 nm. In the numerical analysis, for much smaller particles (24 nm) the evaluated slip length is less than 4 nm.

Difficulties arising with proper interpretation of available measurements of slip length performed by particle tracking became partly understandable if we look at inherent factors modifying mobility of nano-objects. We discuss it in the following chapter.

3. Mobility of nanoparticles

Micro and nano scale motion is coupled or sometimes mainly driven by molecular diffusion, direct effect of molecular structure of our environment. Diffusion governed by Brownian motion is an efficient transport mechanism on short time and length scales. Even a highly organized system like a living cell relies in many cases on the random Brownian motion of its constituents to fulfil complex functions. A Brownian particle will rapidly explore a heterogeneous environment that in turn strongly alters its trajectory. Thus, detailed information about the environment can be gained by analysing the particle’s trajectory. For such analysis spatial resolution down to the nanometre scale is needed. High resolution is directly connected to the requirement to observe the motion on short time scales. However, at short time scales, the inertia of the particle and the surrounding fluid can no longer be neglected, and one expects to see a transition from purely diffusive to ballistic motion [13]. The effect is not negligible for transport phenomena observed in nanoscales, e.g. single-molecule reactions which are basis for transcription of encoded in DNA information. Thus, for complete understanding, an analysis of Brownian motion at very short time scales is necessary, taking effects of inertia into account.

Biochemical reactions in living systems occur in media of very high molecular concentration. In fact it is difficult to talk about diffusion of molecules, at such crowded environment the interactions between macromolecules hinder their displacements, limiting transport and signalling functions [14]. Observation of Brownian motion of micro-objects is classical basis for particles size measurements, evaluation of liquid properties (viscosity, micro rheology), analysing particle – wall interactions, and many others. Nevertheless even such seemingly simple problem creates plethora of uncertainties. In all applications it is necessary to be able to maintain the colloidal well dispersed and to avoid the formation of aggregates. Moreover, it is absolutely necessary to know the fluid-solid interaction in nanoscale and the hydrodynamic properties of the particles.

The equilibrium state and the hydrodynamic properties of many colloids system in aqueous medium is affected to several environmental parameters. The ionic character of water solutions needs beside analysis of hydrodynamic friction (famous Stokes formulae) evaluation of wall interactions. The evaluation of surface charges, ionic streams, creation of the electrostatic double layer theoretically is possible with help of Derjaguin-Landau-Verwey-Overbeek (DLVO) theory [15]. Practically we are far from incorporating all necessary molecular and ionic interaction to our macro hydrodynamics, hence commonly used empirical expression called “hydrodynamic diameter”, effectively shadows our lack of knowledge. In several cases such simplification is sufficient for chemical-engineering; it becomes unacceptable if size of particles strongly decreases. Any ionic layer, streams of ions attaching particle, steric interactions with suspended molecules, effectively decrease particle mobility. To predict such effects is crucial for understanding transport processes at the single cell level.

Figure 3 illustrates our attempts [16] to evaluate size effect for Brownian nano particles. It is obvious that decreasing their size, the effective (hydrodynamic) diameter strongly affects their diffusion.

Figure 3: Relative diffusion coefficient measured for polystyrene nano spheres suspended in water and four different solutions of KCl

Detailed experimental analysis of interactions of liquid molecules and surface molecules of individual particle is very difficult. Hence, in practice more or less sophisticated hydrodynamic models are implemented to interpret observed variation of the apparent particle diameter (in fact friction coefficient). Such models used later for measuring and sorting macromolecules are in common use, despite questionable theoretical background given by fluid mechanists.
Recently, a new optical tool, so called Optical Tweezers (OT) expanded our traditional instrumentation creating possibility for undisturbed measurements of forces and position in picoNewton and nanometre scales. Dragging, towing single particle allows to perform precise analysis of forces involved by liquid environment, wall interactions, and particle-particle interactions. One of the fundamental problems of single particle mobility, namely ballistic regime and effects of inertia creating time dependent recirculation of surrounding liquid molecules, could be proven using OT [17]. Electronic way of signal analysis allows for thermal motion of particle trapped by OT to be evaluated with MHz sampling frequency and displacements below 1 nanometre. In our preliminary study [18] OT developed at IPPT have been used to analyse Brownian motion of trapped polystyrene particle. It appears that already at sampling times of 10kHz diffusion becomes influenced by ballistic regime of molecular interactions (Fig. 4).

Figure 4: Stiffness of the Optical Tweezers evaluated for 1µm polystyrene particle suspended in water; particle diffusion increases as the mean Brownian displacement decreases (upper axis). Straight line - large displacements theoretical limit

4. Worm-Like Chain (WLC)

The flow of deformable objects (fibres, polymer chains) has non-Newtonian character, strongly influencing its short time response at microscopic level [19,20]. Under flow these objects are oriented, deformed, and coiled leading to a macroscopic variation of the transport properties. The microscopic structures, as well as the macroscopic response, depend on both the nature of the suspended objects and the flow configuration. Linking mechanical and microscopic properties of the suspended objects to the macroscopic response of the suspension is one of the fundamental scientific challenges of soft matter physics and remains unsolved for a large number of situations typical for intercellular transport of proteins and ligands.

Most of the biomolecules have strongly elongated form, far from idealistic ball like shape. Their penetration through the crowded cellular environment is strongly enhanced by its shape flexibility. The interplay between crowding and thermal bending allows for the controlled mobility.

For micro or nanoscale objects their natural shape deformation are induced by thermal fluctuation leading to some intriguing behaviour. For example, it has been predicted that long fibres may perform spectacular windings to form more or less stable knots, phenomenon of fundamental importance for biological macromolecules [21].

Despite the strong recent expansion of this field, it still lacks experimental investigations to validate the assumptions of the theoretical and numerical models. The lack of experimental studies is mainly due to the absence of good model systems that allow determining and controlling elasticity and geometry of analysed objects.

Observation of proteins or DNA is still mostly qualitative, limits of the optical methods permit to find out some predicted characteristics only. Therefore, to systematically investigate the influence of long molecules on their interaction with given flows and the resulting macroscopic properties, a synthetic models of flexible objects are useful. Hence, we aimed to produce flexible nanofibres to mimic behaviour of long chains of microparticles. Alas, winding of the objects predicted by simple Stokesian model could not be confirmed [22]. It was probably due too high stiffness of nanofibres used in the experiment.

Recently, we have developed new method allowing constructing highly deformable microscopic filaments with typical diameter of 100nm and contour length ranging from single micrometres to millimetres [23] (Fig. 5). Introducing them directly to microfluidic channels allowed us to observe their deformations due to the flow as well as those induced by thermal fluctuations. The last effect has additional advantage, analysis of thermal fluctuations of flexible objects is used to evaluate their persistence length [24], directly correlated with its mechanical properties. Typical values of persistence length obtained for our hydrogel filaments range from 5 nm to 70 nm, being very close to that reported for DNA chains (30nm).

Figure 5: Hydrogel nano-filament observed under an atomic force microscope; contour length 7 µm, diameter 80nm [23]
Hence, we are able to validate worm-like chain models with material parametrization obtained from the experiment. This approach could in the future be used to gain further fundamental understanding of filaments dynamics under flow as a function of their complex properties as anisotropy or deformability. It is interesting to note that even for relatively low flow Reynolds number (Fig. 6) we observed typical coiling – uncoiling sequences. It is remarkable similar to WLC modelling performed with Stokesian approach [21,22]. Understanding the link between the microscopic structure of the filaments and the macroscopic flow properties opens the possibility to design nano-objects transported by body fluids for targeted drug release or local tissue regeneration.

5. Conclusions

Recent development of experimental techniques applicable to fluid mechanics of nano and microscale permits to have a closer look at applicability of existing mechanical models to small scale phenomena. In the following we have presented few selected problems characterizing nano and microscale fluid dynamics, namely kinematic boundary condition at solid interfaces and suspension of nano scale intrusions, like nanoparticles and long, deformable filaments. In modelling of such nanoscale problems, similarly to solid mechanics science, there is still unsolved problem of merging atomistic scales with nano, micro and macro systems. In case of fluid mechanics there is additional difficulty, time stepping merging has to be performed for very short time scales. It is challenging, still not available approach. Therefore, we have to cope with simplified models, where solely experimental validation may offer background for tuning and adjustment of crucial model parameters.

References


Experimental attempts for creep and fatigue damage analysis of materials – state of the art and new challenges

Zbigniew L. Kowalewski
Institute of Fundamental Technological Research, Polish Academy of Sciences
Pawińskiego 5B, 02-106 Warsaw, Poland
e-mail: zkowalew@ippt.pan.pl

Abstract

Development of creep and fatigue damage was investigated using destructive and non-destructive methods in materials commonly applied in power plants or automotive industry. In order to assess such kind of damage the tests for a range of different materials were interrupted for selected time periods (creep) and number of cycles (fatigue). As destructive methods the standard tension tests were carried out after prestraining. Subsequently, an evolution of the selected tension parameters was taken into account for damage identification. The ultrasonic and magnetic techniques were used as the non-destructive methods for damage evaluation. In the final step of the experimental programme microscopic observations were performed.

Keywords: creep, fatigue, damage, non-destructive methods

1. Introduction

Many testing techniques commonly used for damage assessments have been developed up to nowadays. Among them one can generally distinguish destructive and non-destructive methods. Having the parameters of destructive and non-destructive methods for damage development evaluation it seems to be reasonable their further analysis that should provide possible mutual correlations [1]. This is because of the fact that typical destructive investigations, like standard tests, give the macroscopic parameters characterizing the lifetime, strain rate, yield point, ultimate tensile stress, ductility, etc. without sufficient knowledge concerning microstructural damage development and material microstructure variation. On the other hand, non-destructive methods provide information about damage at a particular time of the entire working period of an element, however, without sufficient knowledge about the microstructure and how it varies with time. Therefore, it seems reasonable to plan damage development investigations in the form of interdisciplinary tests connecting results achieved using destructive and non-destructive methods with microscopic observations in order to find mutual correlations between their parameters. This is the main issue considered in this paper.

The microscopic observations of microstructural changes carrying out using traditional scanning electron microscopes are possible only after specimen failure, and additionally they suffer on high cost. The non-destructive methods are much more convenient and that is why they are more frequently used in engineering practice for periodic evaluation of the material degradation.

The resolution range of particular non-destructive tests covers the entire scope of effects and crystallographic defects [2], Fig. 1. Considering the detailed conditions and scopes of applying various methods significantly limits the possibilities to use them and makes it difficult to identify and analyse the fatigue damage evolution in the experimental way. This leads to a necessity of the constant improvement of existing non-destructive testing methods and of developing new measurement techniques that would be able to detect and carry out quantitative assessment of the structural failures resulting from the development of processes leading to material fatigue and deterioration of its mechanical properties.

Figure 1: Dimensions of microstructural forms/defects and non-destructive methods suitable to monitor them [2]

The conventional non-destructive techniques (based on the measurement of ultrasonic wave velocities for example) are sensitive to a material damage in the last stage of material life and do not give the answer into the question when a given
element should be withdrawn from the exploitation? Therefore, there is the need to develop a new method based on a single or combination of non-destructive techniques allowing to estimate the microstructural and mechanical degradation observed at different stages of the exploited material.

2. Experimental attempts for creep damage analysis

2.1. Multiaxial creep tests

The results from uniaxial creep tests are not able to reflect complex material behaviour. Therefore, many efforts are focused on tests carrying out under multiaxial loading conditions. The well known method of creep rupture data from such tests analysis is through isochronous stress-strain curves obtainable from the standard creep curves [3]. It gives comprehensive graphical representation of material lifetime.

The curves of the same time to rupture determined on the basis of experimental programme are compared in Fig.2 with theoretical predictions of the three well known creep rupture hypotheses: (a) the maximum principal stress rupture criterion (Eqn 1), (b) the Huber-Mises effective stress rupture criterion (Eqn 2), (c) the Sdobyrev creep rupture criterion (Eqn 3). For the biaxial stress state conditions, realised in the experiment (Eqn 2), (c) the Sdobyrev creep rupture criterion (Eqn 3). For the biaxial stress state conditions, realised in the experimental programme, the rupture criteria mentioned above are defined by the following relations:

\[
\sigma_b = \sigma_{\text{max}} = \frac{1}{2}\left(\sigma_{11} + \sqrt{\sigma_{11}^2 + 4\sigma_{12}^2}\right)
\]  
\[
\sigma_b = \sigma_c = \sqrt{\sigma_{11}^2 + 3\sigma_{12}^2}
\]  
\[
\sigma_b = \beta \sigma_{\text{max}} + (1 - \beta)\sigma_c
\]  

Figure 2: Comparison of the isochronous creep rupture surfaces \((\tau_r = 500 \; [\text{h}])\) determined for aluminium alloy (1 - experimental results; 2, 3, 4 - theoretical predictions using the maximum principal stress criterion; the effective stress criterion; and the Sdobyrev criterion, respectively)

As it is clearly seen, the best fit of the aluminium alloy data is obtained using the effective stress rupture criterion. It has to be noting however, that the lifetimes predicted by this criterion are still quite far from experimental data.

2.2. New concepts of creep analysis

To assess damage using destructive method the specimens after different amounts of prestraining were stretched to failure [1, 4]. Afterwards, the selected tension parameters were determined (yield point, ultimate tensile stress) and their variations were used for identification of damage development. Ultrasonic and magnetic investigations were selected as the non-destructive methods for damage development evaluation. For the ultrasonic method, the acoustic birefringence coefficient was used to identify damage development in the tested steel. In the case of magnetic method a several damage sensitive parameters were identified, e.g. amplitude of Ub or Ua envelopes reflecting the Barkhausen effect (HBE) or magneto-acoustic emission (MAE) variation, respectively, their integrals, and coercivity.

Having selected parameters of destructive and non-destructive techniques, possible relationships between them were evaluated. The representative relationships are illustrated in Fig. 3a, b, c. As it is seen, except of the specimen prestrained up to 10.5% due to plastic flow, all results are ordered, and as a consequence, they can be well described by adequate functions depending on the type of prior deformation.

Diagram (a) in Fig. 3 shows relationship between integral of the Ub envelope amplitude - Int(Ub) and ultimate tensile stress. The magnetic parameter is normalized to the value captured for the non-deformed specimen. Numbers in figure denote the level of prior deformation. Figure 3a allows concluding that integral \(\text{Int}(U_b)\) of the Barkhausen noise may be used to estimate a level of the ultimate tensile stress of plastically deformed specimens. It has to be noticed however, that in the case of the material prestrained due to creep the situation is more complicated, since a non-unique relationship between \(R_m\) and \(\text{Int}(U_{b,\text{hom}})\) was found.

The relations in Fig. 3a indicate that the steel after plastic deformation, leading to higher values of \(R_m\), can be characterised by lower values of magnetic parameters. This is because the prestrained material contains more dislocation tangles that impede domain walls movement. On the other hand, the lower values of magnetic parameters can be attributed to the lower magnitudes of \(R_m\) for the steel after creep.

The results make evident that the MBE intensity varies significantly due to microstructure modification, however, in different ways depending on prior deformation type. For deformation higher than 2% this intensity decreases after plastic flow and increases after creep. Strongly non-linear character of plots in Fig. 3a makes impossible direct estimation of mechanical parameters when only single magnetic parameter is used. Addressing the issue for practical application of the MBE measurement in assessment of mechanical properties for the damaged steel one can conclude that it is possible only then if at least two magnetic parameters can be attributed to the lower magnitudes of \(R_m\) for the steel after creep.

Better correlation was achieved between \(R_m\) and coercivity \(H_c\), Fig. 3b. As it is seen, except specimens prestrained up to 10.5% due to plastic flow, all results are ordered, and as a consequence, they can be well described by adequate functions depending on the type of prior deformation. The main disadvantage of the relationships between \(R_m\) and \(H_c\) is related to the fact that it cannot distinguish a type of prior deformation for small prestrain magnitudes.

Similar remarks can be formulated for the relationships between \(R_m\) and acoustic birefringence coefficient \(B_{\text{BA}}\), Fig. 3c.
to provide more thorough analysis reflecting physical interpretation of the relationships obtained further investigations are necessary. Programmes of such tests should contain advanced microscopic observations using not only optical techniques, but also SEM and TEM.

3. Experimental attempts for fatigue damage analysis

The fatigue of structural materials is particularly important for the development of railway and air transport as well as power engineering. Fast introducing of new technologies in those sectors frequently leads to serious crashes. It is enough to mention two crashes of the first jet-propelled passenger aircraft called Comet, crash of the fast German railway in Eschede in 1998 or two jet crashes of the first transoceanic airway line in Poland in 1980 (Okęcie Airport in Warsaw) and in 1987 (Kabaty near Warsaw). All those tragic crashes were caused by the material fatigue, the nature of which has not been examined sufficiently.

Experimental determination of damage development requires on-line recording the material responses to the given cyclic loading during the whole experiment. Difficulty in conducting of this task is strengthened additionally by a lack of the effective fatigue damage measure describing properly effects of material degradation. Failure mechanics is relatively a new field of studies and the reference fatigue damage indicators described in scientific works should be treated only as suggested, but not verified and accepted problem solution. Monitoring of the changes in mechanical properties taking place under cyclic loads by means of recording stress and deformations of the measured sample section in consecutive cycles was an efficient technique enabling not only to define the number of cycles required to damage the specimen, but also to define the fatigue damage indicator and its evolution in the fatigue process.

Behaviour of materials within the range of high cycle fatigue (HCF), which means the stress amplitude is below the yield limit of a material, can be divided into two basic types in terms of mechanisms of damage development [5]. Behaviour of the first group of materials is described by the ratcheting generated by local deformation around voids, non-metallic precipitations and other defects of structure. Behaviour of the second group of materials undergoing cyclic loading is described by cyclic plasticity generated by slips on the level of grains and local slip bands. In both cases the changes of measured deformations are a sum of local deformations in the first group of materials is described by the ratcheting and as a consequence, its development in consecutive loading cycles. Their imaging (Figs. 4 and 5) helps not only to define the damage development mechanism, e.g. in the form of debonding or decohesion, but also provides the basis for modelling of their mechanisms development.

The metallographic methods are laborious and problematic due to the specific conditions of sample preparation. Usually they are mainly used after completion of the loading process or after its programmed suspension in the desired phase. Their significant limitations are due to the laborious procedures of sample preparation and the necessity to observe them after load removal. A few years ago, a unique research station has been constructed, with a servo-hydraulic testing machine equipped with digitally controlled signals of load, displacement, total or plastic strain, which can be used for monotonic stretching and compressing of specimens in the range of ±100 kN and for low-cycle fatigue tests (LCF) in temperature range up to 1100°C.

Figure 3: Variation of ultimate tensile stress of the X10CrMoVNb9-1 steel versus: (a) integral of the Ub envelope amplitude; (b) coercivity; (c) birefringence coefficient (triangles – steel after creep; circles – steel after plastic flow)

The relationships between selected destructive and non-destructive parameters sensitive for damage development show a new feature that may improve damage identification. In order to provide more thorough analysis reflecting physical interpretation of the relationships obtained further investigations are necessary. Programmes of such tests should contain advanced microscopic observations using not only optical techniques, but also SEM and TEM.

3. Experimental attempts for fatigue damage analysis

The fatigue of structural materials is particularly important for the development of railway and air transport as well as power engineering. Fast introducing of new technologies in those sectors frequently leads to serious crashes. It is enough to mention two crashes of the first jet-propelled passenger aircraft called Comet, crash of the fast German railway in Eschede in 1998 or two jet crashes on the first transoceanic airway line in Poland in 1980 (Okęcie Airport in Warsaw) and in 1987 (Kabaty near Warsaw). All those tragic crashes were caused by the material fatigue, the nature of which has not been examined sufficiently.

Experimental determination of damage development requires on-line recording the material responses to the given cyclic loading during the whole experiment. Difficulty in conducting of this task is strengthened additionally by a lack of the effective fatigue damage measure describing properly effects of material degradation. Failure mechanics is relatively a new field of studies and the reference fatigue damage indicators described in scientific works should be treated only as suggested, but not verified and accepted problem solution. Monitoring of the changes in mechanical properties taking place under cyclic loads by means of recording stress and deformations of the measured sample section in consecutive cycles was an efficient technique enabling not only to define the number of cycles required to damage the specimen, but also to define the fatigue damage indicator and its evolution in the fatigue process.

Behaviour of materials within the range of high cycle fatigue (HCF), which means the stress amplitude is below the yield limit of a material, can be divided into two basic types in terms of mechanisms of damage development [5]. Behaviour of the first group of materials is described by the ratcheting generated by local deformation around voids, non-metallic precipitations and other defects of structure. Behaviour of the second group of materials undergoing cyclic loading is described by cyclic plasticity generated by slips on the level of grains and local slip bands. In both cases the changes of measured deformations are a sum of local deformations in the whole volume being measured. Cyclic loading in the range of HCF causes initiation of various mechanisms of damage development.

Initial defects in the form of inclusions initiate a localisation of damage, and as a consequence, its development in consecutive loading cycles. Their imaging (Figs. 4 and 5) helps not only to define the damage development mechanism, e.g. in the form of debonding or decohesion, but also provides the basis for modelling of their mechanisms development.

The metallographic methods are laborious and problematic due to the specific conditions of sample preparation. Usually they are mainly used after completion of the loading process or after its programmed suspension in the desired phase. Their significant limitations are due to the laborious procedures of sample preparation and the necessity to observe them after load removal. A few years ago, a unique research station has been constructed, with a servo-hydraulic testing machine equipped with digitally controlled signals of load, displacement, total or plastic strain, which can be used for monotonic stretching and compressing of specimens in the range of ±100 kN and for low-cycle fatigue tests (LCF) in temperature range up to 1100°C.
The strength testing machine is connected to the set of scanning electron microscopes for examining the sample surface under monotonic load and microscopic observations with the resolution of $\mu m$. The second electron microscope enables an examination of the specimen surface under constant load, and microscopic observations with the nano-metric (nm) resolution.

The specimen, after fixing and imposing the planned range of fatigue loads, is descended, together with the stress-testing machine frame, into a large vacuum chamber with the capacity of $2m^3$ where the metallographic tests of the entire sample surface are carried out. The device is equipped also with EDX system (Energy Dispersive X-Ray) enabling to carry out local chemical analysis, and with EBSD system (Electron Back Scatter Diffraction) to image the metallographic orientation in the tested field. The first research station of such a type was installed in 2009 at the Institute for Material Science of the Erlangen-Nuernberg University in Germany.

Among many fatigue testing programmes one can distinguish two basic directions: (a) investigations conducted by physicists and metallurgists focusing on trying to learn the mechanisms governing the process of fatigue; (b) theoretical and experimental investigations in order to create a phenomenological theory to allow quantitative description of the phenomenon. Both of these trends are currently developing in parallel.

### 3.1. Fatigue damage evaluation

In many cases the process of fatigue damage is controlled by more than one mechanism. The fatigue tests carried out on metallic materials and metal matrix composites have shown that the damage process occurred due to combination of cyclic plasticity and ratcheting mechanisms. Therefore, using adequate damage indicators, Fig. 6, damage measure can be defined by the following relationship:

$$\psi_N(\varepsilon_m, \varepsilon_m^*) = \sum_{i=1}^{N} \varepsilon_m^* + \sum_{i=1}^{N} \varepsilon_m$$  \hspace{1cm} (4)

Hence, a definition of damage parameter takes the form:

$$D = \frac{\psi_N(\varepsilon_m)_{\text{max}} - \psi_N(\varepsilon_m)_{\text{min}}}{\psi_N(\varepsilon_m)_{\text{max}} - \psi_N(\varepsilon_m)_{\text{min}}}$$  \hspace{1cm} (5)

where $\varepsilon_m$ – accumulated strain up to the current loading cycle, $\psi_N(\varepsilon_m)_{\text{max}}$ – accumulated strain at the first cycle, $\psi_N(\varepsilon_m)_{\text{max}}$ – accumulated total strain calculated for all cycles, in which damage measure is included in the form of equation (4).

![Figure 6: Illustration of strain damage indicators during fatigue conditions](image)

The results published so far [e.g. 6, 7] confirm the correctness of the adopted methodology for damage analysis of the materials after service loads, that taking into account parameters responsible for cyclic plasticity and ratcheting.

### 3.2. Previous and new concepts of fatigue testing

In order to assess damage degree due to fatigue of the material in the as-received state and after exploitation the Wöhler diagrams may be elaborated that represent the number of cycles required for failure under selected stress amplitude. The results of such approach are illustrated in Fig. 7 for the 13HMF steel. As you can see, the Wöhler diagrams depending on the state of material differ themselves, thus identifying the fatigue strength reduction due to the applied loading history. Unfortunately, such method of degradation assessment of the material undergoing fatigue suffers on very high cost and additionally it is time consumable.

![Figure 7: Wöhlers diagram for the 13HMF steel before (0h) and after exploitation (80 000h and 144 000h)](image)

Therefore, searches are conducted for new solutions that would provide a better assessment of the fatigue damage development. In order to obtain this effect, an adequate damage parameter must be defined on the basis of the measurable indicators of its
development. Selected proposals discussed in section (3.1) have been validated experimentally and the representative results will be presented here.

Force controlled high cycle fatigue tests (20 Hz frequency) were carried out on the servo-hydraulic testing machine MTS 858. During the tests, sine shape symmetric tension-compression cycles were applied to keep constant stress amplitude equal to 70 MPa and 350 MPa for metal matrix composite (Al/SiC) and X10CrMoVNb9-1 steel, respectively. Tests were performed at ambient temperature. Each cylindrical specimen manufactured from both materials was subjected to cyclic loading until fracture. A movement of the subsequent hysteresis loops along the strain axis was observed with an increasing number of cycle (Figs. 8 and 9). Simultaneously, a width of the subsequent hysteresis loops became almost unchanged for composite and became greater for the steel. Such behaviour identifies the ratcheting effect.

Since ratcheting is the dominant mechanism of the composite deformation, the mean strain was taken into account during a damage parameter calculation in the stable growth period. Hence, the damage parameter can be defined using equation (4) assuming that its first term is neglected. It is worth to noticed that the rate of damage is relatively high at the beginning of the period. Afterwards, it becomes slower (Fig. 10). In the case of X10CrMoVNb9-1 steel besides of ratchetting also cyclic plasticity mechanism is responsible (increase of hysteresis loop width) for damage development, Fig. 9. Taking into account both mechanisms a variation of the damage parameter can be calculated using equations (4) and (5). In graphical form it is shown in Fig. 11.

Looking at the diagrams in Figs. 10 and 11 one can say that the linear damage accumulation rule cannot be applied for both tested materials. Moreover, it can be seen that the rate of damage parameter variation for composite depends on the SiC particle content. It increases with an increase of the SiC particle content at the initial stage of fatigue, Fig. 10.

The examples of fatigue testing may be treated as an alternative method for damage evaluation of materials subjected to cyclic loads. Studies in which a variation of the hysteresis loop width and its movement were recorded for cycles under fixed constant stress amplitude have demonstrated that this procedure gives a possibility to assess safe operation period for materials in question and there is no need to perform so many experiments, as it is required for the Wöhler diagram determination. The proposed method of assessing fatigue damage evolution makes it possible to: determine damage indicators; define damage parameter; assess fatigue and stress levels to find ranges in which an accumulation of damage can be described by the linear law.

Another very promising attempts in the fatigue damage analysis are related to relatively new techniques such as Electronic Spackle Pattern Interferometry (ESPI) or Digital Image Correlation (DIC). Some preliminary results of the ESPI usage will be presented here. The cast aluminum alloy AlSi7MgCu0.5 for automotive engine heads [6] was tested after exploitation cycles.

Opportunity for strain/stress components measurement of a specimen using ESPI is presented in Fig. 12. The specimen was cut from automotive engine head (cast aluminum alloy AlSi7MgCu0.5). Such heads were cast according to the standard procedure that ensures degassing. The ratio of the
average porosity was 6%. Observing the strain and stress distributions in Figs. 12 and 13 a serious problem can be noticed, i.e. an averaging of the accuracy of the strain components in the entire volume of the geometrically homogeneous specimen. In this group of materials the development of fatigue damage takes place around various drawbacks, mainly in the form of voids formed in manufacturing processes such as casting. Besides of density and distribution of defects in the volume of tested specimen, the specimen size, and location of individual defects are important factors of the fatigue damage initiation and further development. Development of local strain components around defects leads to ratcheting, i.e. the incremental rise of the strain components being in agreement with acting stress direction. This group includes not only all cast alloys, but also the whole range of modern metal matrix composites.

Figure 12: Strain component distribution map for \( z \) direction on the plane specimen surface (cross section 18-mm \( \times 4 \)) under load of 1.2 kN

Figure 13: Transversal distribution of the stress component along \( z \) axis

Solving problems related to development of fatigue damage, degradation of the mechanical properties of structural materials, as well as modeling of the mechanical properties necessary to simulate the behaviour of the structure under loading is based mainly on experimental data from tests under uniaxial stress states. However, for such purposes also data coming from complex stress state experiments are required. The testing technique under complex stress states gives full information about the mechanical properties of structural materials, necessary for parameters determination during numerical modeling. In this context either ESPI or DIC are very helpful since monitoring of the phenomena related to the fatigue, as the process initiating locally, requires the full-field observations of the displacement brought by cyclic loading. Both these methods enable determination of the displacement distribution on the specimen surface, and thus, also strain and stress concentration spots resulting from the defect.

4. Conclusions

The results clearly indicate that selected ultrasonic and magnetic parameters can be good indicators of material degradation and can help to locate the regions where material properties are changed due to prestraining.

Creep or fatigue analysis should be based on the interdisciplinary tests giving a chance to find mutual correlations between parameters assessed by classical macroscopic destructive investigations and parameters coming from the non-destructive experiments. Such relationships should be supported by thorough microscopic tests, giving as a consequence, more complete understanding of the phenomena observed during damage development.

Application of the full-field observation methods (ESPI, DIC) enables localization of the initial spot of the failure resulting from the cyclic stress and monitoring of its development as well.

References


Shape Memory Materials and Structures: Modelling and Computational Challenges

Mieczysław Kuczma*
Faculty of Civil and Environmental Engineering, Poznan University of Technology
Piotrowo 5, 61-960 Poznań, Poland
e-mail: mieczyslaw.kuczma@put.poznan.pl

Abstract

In this presentation we will discuss thermomechanical and mathematical modelling aspects of shape memory materials and structural elements made thereof. Special attention will be given to nonlinear physically-based models which take into account significant hysteresis as well as to numerical approximation and solution techniques for the corresponding rate boundary value problems. A number of current and future applications of shape memory materials will be mentioned. Our aim is to provide a unified constitutive and mathematical frameworks for a class of shape memory materials and structures. Theoretical considerations will be illustrated with results of numerical simulations.

Keywords: shape memory alloys, shape memory polymers, hysteresis, constitutive modelling, numerical simulation, complementarity problems, variational inequalities

1. Introduction

Shape memory is the uncanny ability of a material to return to its original configuration from seemingly permanent deformation after removing loading and eventually by heating above a certain characteristic temperature. This intriguing phenomenon is exhibited by some classes of materials, including shape memory alloys (SMAs) and shape memory polymers (SMPs) [10, 8]. SMAs have been used as active materials in smart structures and various sophisticated devices due to their actuation, damping and sensing properties for more than three decades now, and recently they have also found applications in civil and structural engineering [13].

The unusual behaviour of SMAs, including the shape-memory effect and pseudoelasticity, is a result of a crystallographically reversible martensitic phase transformation. It is a first order solid-solid phase transition with displacive nature from a highly ordered parent phase (austenite) to a less-ordered product phase (martensite) and the reverse one, which can be induced by changes in stress and/or temperature (or magnetic field). Pseudoelasticity (named also superelasticity) of a SMA means that the material is capable to sustain large deformations (8-10%) and to retain its non-deformed shape upon unloading, at temperatures above \( T_f \) (austenitic finish temperature).

A number of classes of SMA models can be distinguished but usually, models aimed at solving realistic boundary value problems are based on continuum thermomechanics and make use of internal variables to account for the microstructural changes due to the martensitic phase transformation, e.g. [1, 9, 3, 12, 4, 2, 5, 14, 7, 11, 6], to cite a few works.

2. Pseudoelastic hysteresis

A characteristic feature of the pseudoelastic behaviour of SMAs is a flag-type hysteresis schematically shown in fig. 1. This pertains to the 1D case with austenite \( A \) and two variants of martensite \( M_1 \) and \( M_2 \), and 1D-measures \( \eta_1 \) and \( \eta_2 \) of their phase transformation strains, respectively.

\[
X = \begin{cases} \frac{c - c_m^+}{c_m^+ - c_m^-} & \text{if } \eta < 0 \\ \frac{c - c_m^0}{c_m^+ - c_m^-} & \text{if } \eta > 0 \end{cases}
\]

\[
\dot{c} = \begin{cases} \frac{\dot{c} - \dot{c}_m^0}{\dot{c}_m^+ - \dot{c}_m^-} & \text{if } \eta < 0 \\ \frac{\dot{c} - \dot{c}_m^0}{\dot{c}_m^+ - \dot{c}_m^-} & \text{if } \eta > 0 \end{cases}
\]

In describing the main loops (bold lines in fig. 1) we make use of the phase transformation driving forces \( X_m \) \((m = 1, 2)\) and some threshold functions \( \kappa_m^- \) and \( \kappa_m^+ \) for positive and negative rates of change of volume fractions of martensitic variants \( c_m \), respectively. The phase transformation rules can be expressed as follows

\[
\begin{align*}
X_m(c_m) &= \kappa_m^+(c_m) & \text{then } \dot{c}_m \geq 0 \\
X_m(c_m) &= \kappa_m^-(c_m) & \text{then } \dot{c}_m \leq 0
\end{align*}
\]

Condition (1) controls the forward phase transformation of \( m \)-th variant of martensite and (1)2 — the reverse one, whereas (1)3 means that the deformation process is pure elastic. The threshold functions \( \kappa_m^- \) and \( \kappa_m^+ \) are measures of the energy dissipated in the course of forward (\( \dot{c}_m \geq 0 \)) and reverse (\( \dot{c}_m \leq 0 \)) phase transformation.

*Work carried out under the grant no. 01/11/DSPB/0400 at the Institute of Structural Engineering, Poznan University of Technology.
3. Thermomechanical constitutive relations

We consider a material that may appear in \( N + 1 \) preferred strain states: the parent phase (austenite), indexed with \( i = N + 1 \equiv a \), and \( N \)-variants of martensite (\( m = 1, \ldots, N \)). The Helmholtz free energy \( W_i \), \( i = 1, 2, \ldots, N + 1 \) is postulated in the form

\[
W_i(\varepsilon, \theta) = \frac{1}{2} (\varepsilon - d_i) \cdot A_i [\varepsilon - d_i] + \phi_i(\theta),
\]

where \( \varepsilon \) is the total strain tensor, \( A_i \) is the elasticity tensor and \( d_i \) the transformation strain of \( i \)-th phase (variant), and \( \phi_i(\theta) \) is a stress-free part dependent on temperature \( \theta \).

The relaxed form of multi-well free energy function with quadratic wells (2) of the austenite-martensite mixture may be defined as

\[
\tilde{W}(\varepsilon, \theta, c) = \frac{1}{2} (\varepsilon - d(c)) \cdot A [\varepsilon - d(c)] + \phi(\theta, c) + \psi_{mix}(c)
\]

where \( c = \{ c_i \} \) is a column matrix that gathers volume fractions of martensite variants and austenite \( c_i, \psi_{mix} \) represents interfacial energy in the mixture, and the effective transformation strain \( d(c) \) is the convex combination,

\[
d(c) \equiv \sum_{i=1}^{N+1} c_i d_i, \quad \text{with} \quad c_i \geq 0, \quad \sum_{i=1}^{N+1} c_i = 1.
\]

From the reduced form of the dissipation inequality

\[
\frac{D}{\partial c} \varepsilon \equiv X \cdot c = \sum_{m=1}^{N} X_m \cdot \dot{c}_m \geq 0
\]

one obtains components \( X_m \) of the phase transformation driving force,

\[
X_m = d_m \cdot A [\varepsilon] - \sum_{i=1}^{N} (d_m \cdot A [d_i] + B_{nm}^s c_i - (\alpha_m - \alpha_n) - B_{nm})
\]

4. Computational model

Due to the inequality constraints (1), (4), (5) the considered problem must be solved incrementally in time, and iteratively in space in case of geometrical nonlinearities.

4.1. Variational inequality

Let \( u \) be the displacement vector of a body occupying a region \( \Omega \subset \mathbb{R}^3 \) and define finite increments in a typical time step \( t_{n-1} \rightarrow t_n \) as \( u_n \equiv u_n - u_{n-1} \) and analogously for the concentration of phases \( c_n \equiv c_n - c_{n-1} \). Let \( V(t_n) \in H^1(\Omega, \mathbb{R}^3) \) be the set of kinematically admissible increments of displacements and \( K_{\pm}(c_{n-1}) \) stand for sets of increments of volume fractions at time \( t_{n-1} \).

The incremental boundary value problem is governed by the variational inequality: Find \( (\hat{u}_n, \hat{c}_n) \in V(t_n) \times K(c_{n-1}) \) such that

\[
a(\hat{u}_n, v) - g(\hat{c}_n, v) = f_{\pm, n-1}(v)
\]

\[
\mp g(z_{\pm} - \hat{c}_n, \hat{u}_n) \pm h(z_{\pm} - \hat{c}_n) \geq \mp h_{\pm, n-1}(z_{\pm} - \hat{c}_n)
\]

for all \( v \in V(t_n), \) \( z_{\pm} \in K_{\pm}(c_{n-1}) \). These conditions reflect the structure of the problem, in which bilinear or linear forms \( a, g, h, f, b \) constitute a weak form of the equilibrium equations (first, variational equation) and a weak form of the phase transformation rules (variational inequality).

4.2. Incremental linear complementarity problem

The finite dimensional counterpart of the variational inequality formulation may be expressed as the following (linear) complementarity problem:

\[
D x_n + y_n = b_{n-1}, \quad x_n \geq 0, \quad y_n \geq 0, \quad x_n \cdot y_n = 0
\]

where \( D \) is a square matrix, \( x_n \) is a vector of unknowns (nodal values of the finite element approximations), \( y_n \) denotes a vector of slack variables, and the vector \( b_{n-1} \) is known at time \( t_{n-1} \).

5. Closing remarks

The developed formulation of the pseudoelastic deformation process takes into account hysteretic effects and automatically determines the moving phase transformation front. The formulation of this free boundary value problem becomes further complicated when one takes into account the fact that the austenite-to-martensite phase transformation is exothermic whereas the reverse martensite-to-austenite one is endothermic. The moving source of heat and sink of heat render the considered deformation process rate dependent through a coupled heat equation. Results of numerical simulations for various structural elements (bars, trusses, beams, plates) as well as projected future applications of shape memory materials will be presented.

References

Advanced mechanics in High Energy Physics experiments

Tadeusz Kurtyka
CERN – European Organization for Nuclear Research
CERN, CH-1211 Geneva 23, Switzerland
e-mail: Tadeusz.Kurtyka@cern.ch

Abstract

Research in High Energy Physics (HEP) is now based on a world-wide collaboration, but is not restricted to only physicists; the challenges of construction of giant instruments of this research, like the Large Hadron Collider, often at the cutting-edge of available technologies, require advances and developments in other domains of science and technology. This includes also a broad class of fields of mechanics where such advances are needed, encompassing new theoretical developments, novel experimental techniques and more efficient computational tools. Large HEP laboratories, and CERN in particular, rely here on collaboration with a network of external research centres capable of providing such an advanced mechanics expertise. This network includes already some Polish partners; future projects of HEP will require a still wider collaboration in various fields of mechanics. The present paper gives an overview of some of these fields, and may be treated as an invitation to the Polish Mechanics community for broadening this network.

Keywords: particle accelerators, mechanics, interaction of particle beams with matter, beam intercepting devices, materials at cryogenic temperatures, strain induced degradation of superconductors, active vibration isolation systems

1. Introduction

Research in particle physics relies on innovative tools, technologies, and techniques which depend critically on advances from other fields. The construction of the Large Hadron Collider (LHC), the world’s largest and most powerful particle collider, has required an extensive collaboration of specialists from hundreds of universities and laboratories, and has driven a development work in several domains, including various fields of mechanics. Some of them are briefly described in the present paper which also signals the domains where further collaboration would be highly desirable.

2. Thermo-mechanics of interaction of high energy particle beams with matter

Highly energetic particle beams interact with matter by producing a shower of secondary particles and depositing heat. In the case of short beam pulses and high energy densities this leads to thermally induced dynamic phenomena in the impacted material. Restricting our further attention to the beam impacts on solids, let us note that, historically, this effects were first studied in the context of target-type physics experiments, where deposited beam energies were relatively low and the corresponding mechanical analysis relatively simple [1].

In the collider-type experiments the beam energies are much higher, reaching unprecedented level in the case of the LHC; its two opposite colliding beams of protons carry the energy of up to 724 MJ (equivalent to 173 kg TNT), which must be safely handled by beam intercepting devices, including a system of collimators removing a small part of this energy during normal operation, but designed against higher energies of accidental beam impacts, up to the system of beam-dumps used for discharging the total beam energy at the end of each physics runs (once, twice per day).

The design challenges of the beam intercepting devices for the LHC, exposed to potentially destructive beam impacts, have driven at CERN a vigorous development of the methods of mechanical analysis of such phenomena, supported from the early design stages by an intensive collaboration with external research centres, particularly those specialized in impact studies and in computational mechanics.

At present, a considerable expertise has been gathered allowing analysis of a wide range of material and structural behaviour, covering the three usual dynamic regimes of analysis, namely: 1). Elastic wave regime, which may be effectively treated with the use of standard implicit FEM codes, or even with some analytical solutions based on classical thermo-elasticity [1] and allowing “benchmark” testing of numerical results; 2). Plastic wave regime, where - with certain precaution - implicit FE methods may still be applied, but where analytical solution for these kind of beam induced loads are lacking; 3). Shock wave regime, potentially leading to severe damage or destruction of structural elements, with the propagating shock waves exceeding the elastic sound speed, and with significant discontinuities in pressure, density and temperature. Here, only explicit FEM codes and “Hydrocodes” are applicable.

Effective use of these computational tools, particularly for the third regime, requires an extensive experimental calibration of dynamic yield conditions, failure criteria, and Equations of State (EOS). Here, CERN relies on external collaborators and available databases, quite scarce in the case of the EOS. On the other hand, an important element has recently been added to this experimental base, namely an unique, dedicated HiRadMat test facility at CERN [2], where structures and materials can be tested under real high energy proton beam impacts, and where the results of computational simulations could for the first time be verified, with an astonishingly good agreement (Fig. 1), by tests involving destructive levels of beam energies.

The investment in such a test facility is highly justified by the current design works devoted to the next generation of collimators for the LHC (designed for higher beam energies) and by future HEP projects where the stored beam energies may be higher by one order of magnitude. These new projects call again for collaboration in the development of novel materials, their experimental characterization and constitutive modelling.
which, apart from complex dynamic characteristics, should also account for the effects of the radiation induced damage.

Figure 1: Flow of material fragments from a specimen under proton beam impact; fast-camera measurements, numerical simulation and post mortem inspection [2]

3. Mechanics of materials

The construction of the LHC has required an extensive testing and qualification of structural materials, often specially developed, to meet the requirements of high temperature applications, as discussed above, but also for the cryogenic environment of the LHC superconducting magnets, cooled by superfluid (1.9 K) or liquid (4.2 K) helium.

Mechanical testing of materials at liquid He temperatures has been intensively developed, both at CERN and in collaborating institutes, giving a new inside into materials behaviour at these temperatures and driving in turn development of new constitutive material models. Such phenomena as discontinuous plastic flow (“serrated yielding”), plastic strain induced phase changes and damage effects have found new theoretical formulations, both by extending the classical phenomenological framework of plasticity, but also by demonstrating that for the adequate description of these phenomena a more deep physical, ab initio approach (e.g. based on phonon mechanism), is needed [7,8].

This need is particularly visible in the description of the so called strain induced degradation of superconductors. This effect is now intensively studied, mainly for the Nb3Sn superconductors and some high temperature superconductors foreseen for the next generation of superconducting magnets for HEP experiments. The phenomenon, known for a wide class of superconductors (e.g. A15-phase intermetallics compounds), is observed as a marked decrease of superconducting properties under mechanical strain. In the current engineering practice this effect is accounted for by various empirical scaling laws [3], formulated in terms of the elastic strain tensor invariants, however there are now several attempts to describe this effect on the basis of more physical arguments, e.g. again on phonon mechanism as initiated in [6].

4. Other domains of mechanics

4.1. Dynamics, vibrations

Analysis of vibrations is now nearly routinely performed for accelerator and detector elements, usually requiring extremely high accuracy of positioning and stabilization, and therefore sensitive to mechanical and “white noise” type seismic disturbances. Future linear colliders present here a particularly challenging task, with a necessity of stabilizing some of its elements to the accuracy of nanometres. Active vibration isolation systems are therefore intensively studied and tested, e.g. for the Compact Linear Collider (CLIC) developed at CERN [5], with quite encouraging results.

4.2. Mechanics of fluids

Mechanics of fluids, as applied in the context of HEP experiments, encompasses a wide range of problems, including thermo-hydraulics of liquid and superfluid He, which would require a separate specialist overview. Let us signal here only some analyses and tests which have been essential for the safety of experimental facilities at CERN, including the problem of sudden bursts and propagation of gases in confined spaces [4], or the analysis of pressure wave phenomena following a break of large vacuum volumes.

The analysis of implosions of vacuum systems which occurred during last two decades in some world physics laboratories, e.g. at the Super-Kamiokande neutrino detector, surely deserves a more profound analysis. Finally, let us note that for several future projects of physics, liquid beam targets impacted by high energy proton beams are considered as the only feasible solution to handle extremely high power loads necessary to produce the secondary particles for experiments in neutrino and muon physics or for testing the concepts of Accelerator Driven Systems (ADS) for future sub-critical nuclear reactors. This extends the problems discussed above in Par. 2 to the field of interaction of particle beams with liquids.

References

Hierarchic isogeometric analyses of beams and shells

Bastian Oesterle\(^1\), Manfred Bischoff\(^2\), Ekkehard Ramm\(^3\)*

\(^1,2\)Department of Civil and Environmental Engineering, University of Stuttgart
Pfaffenwaldring 7, 70550 Stuttgart, Germany

e-mail: oesterle@ibb.uni-stuttgart.de, bischoff@ibb.uni-stuttgart.de, ramm@ibb.uni-stuttgart.de

Abstract

The higher inter-element continuity of the Isogeometric Analysis (IGA) applying NURBS functions for geometry as well as mechanics opens up new possibilities in the analysis of thin-walled structures, i.e. beams, plates and shells. The contribution addresses the straightforward implementation of classical theories requiring C\(^1\)-continuity, such as the Euler-Bernoulli beam and Kirchhoff-Love shell theory. Based on these “simplest” models shear deformable theories, introducing Timoshenko and Reissner-Mindlin kinematics, are formulated in a hierarchic manner. In contrast to the usual Finite Element concept using total rotations the present model picks up traditional formulations introducing incremental rotations as primary variables. Furthermore an alternative version is discussed with a split of the displacements into bending and transverse shear parts. Both hierarchic concepts can be easily extended to 3D–shell models. The key aspect of this alternative parameterization is the complete a-priori removal of the transverse shear locking and curvature thickness locking (in the case of 3D-shells).

Keywords: Isogeometric Analysis (IGA), locking, beams, shells, hierarchic formulation, alternative parameterization

1. Introduction

From the different concepts for dimensional reduction of thin-walled structures, in the present study a derived approach is applied utilizing a-priori assumptions across the thickness for selected mechanical parameters. In the early days Finite Elements were mostly derived for transverse shear free formulations, e.g. a Kirchhoff-Love model for plates and shells. It turned out that the C\(^1\)-continuity requirement was a severe obstacle leading to sophisticated discretization schemes. Consequently later on most formulations applied shear deformable theories requiring only C\(^0\)-continuity, e.g. applying Reissner-Mindlin kinematics. This in turn opened the field of locking problems for equal low order interpolation, such as transverse shear, membrane and thickness locking, resulting in an almost never-ending discussion on appropriate unlocking schemes up to the present time, see review article [1].

This situation lead to the objective of the present study: find a modified parameterization for a primal formulation as hierarchic model with equal (low) order interpolation free from locking avoiding any numerical unlocking scheme. It is found that the Isogeometric Analysis [2], [3] using NURBS functions is an elegant basis for the raised task.

2. Hierarchic formulation of beams and shells

2.1. Model case of straight beam

For simplicity the hierarchic concept is explained for straight Timoshenko beams:

\[
\kappa = \gamma \text{'} \quad \text{(1)}
\]

\[
\gamma = w \text{'} + \phi \quad \text{(2)}
\]

Eqn (1) describes the curvature \(\kappa\) and (2) denotes the shear strain \(\gamma\). Instead following the standard way and discretizing the displacement \(w\) and the total rotation \(\phi\), displacement \(w\) and the shear angle \(\gamma\) itself can be introduced as primal parameters:

\[
\kappa = - w \text{''} + \gamma \quad \text{(3)}
\]

\[
\gamma = \gamma \quad \text{(4)}
\]

It is apparent that this alternative discretization yields the second derivative of \(w\), which requires C\(^1\)-continuity of the trial functions. Eqns (3) and (4) can be identified as the kinematic equations of the Euler-Bernoulli beam enriched by a shear strain \(\gamma\); it plays the role of a hierarchic rotation superimposed to the shear-free beam. In the thin limit \(\gamma\) vanishes what can be easily satisfied by (4). Thus this formulation is free from shear locking.

It should be noted though that the unbalance in derivatives in Eqn (2) is now shifted to \(\kappa\). The second derivative in the displacement \(w\) leads to oscillations in the shear force which are in particular pronounced for quadratic shape functions. This disadvantage may be remedied resorting to a different parameterization decomposing \(w\) into a bending part \(w_b\) and a shear part \(w_s\):

\[
w = w_b + w_s \quad \text{(5)}
\]

so that bending and shear are completely decoupled:

\[
\kappa = w_b \text{''} \quad \text{(6)}
\]

\[
\gamma = w_s \quad \text{(7)}
\]

In this version again shear locking does not occur.

2.2. Curved beams and shells

The extension to curved beams and shells can be done in a similar way. Here the starting point is again a transverse shear-free solution (Euler-Bernoulli or Kirchhoff-Love) where the C\(^1\)-continuity is easily satisfied by the NURBS discretization within the IGA concept [4].

Figure 1: Conventional vs hierarchic parameterization
Figure 1 shows the conventional and the hierarchic parameterization depicting the rotation of the initial director $\mathbf{a}_3$ into its deformed state $\mathbf{a}_3$. $\Phi$ denotes the rotation vector according to the Kirchhoff-Love (K-L) formulation. $\gamma$ represents the difference rotations; $w$ is the related difference displacement at the shell surface. Initial position and base vectors, the displacements of the midsurface as well as the hierarchical rotations are discretized by NURBS shape functions in the sense of IGA. The three components of the midsurface displacement (3-parameter K-L formulation) are enhanced by the two hierarchical rotations leading to a shear locking-free 5-parameter formulation with Reissner-Mindlin (R-M) kinematics [5].

The K-L formulation, its extensions to a hierarchical R-M model and the expansion to a 3D-model constitute a family of beam or shell models ideally suited for model adaptation.

3. Conclusions

The hierarchical concept applying incremental instead of total rotations within the IGA framework completely removes transverse shear-locking as well as curvature thickness locking (for 3D-shells) a-priori on the formulation level. However additional research is necessary to also remove the membrane locking in a similar way. Currently this is done either by a HS- or by a DSG-approach.

References


Finite Element Method simulations of linear and non-linear elasticity problems with error control and mesh adaptation

Waldemar Rachowicz
Institute of Computer Science, Cracow University of Technology
ul. Warszawska 24, 30-155 Kraków, Poland
e-mail: wrachowicz@pk.edu.pl

Abstract

The paper presents the main concepts of error estimation and adaptivity for linear and non-linear elasticity problems solved with the Finite Element Method. We discuss possibility of simulations of boundary-value problems in classical linear elasticity and in finite elasticity with nearly incompressible and (possibly) nearly inextensible materials. We present the most popular a posteriori residual error estimation techniques, the goal-oriented error estimation and the rules of adaptivity of meshes.

Keywords: finite element method, error estimation, mesh adaptivity, goal-oriented, hyperelastic material

1. Introduction

Numerical simulations of problems in solid mechanics constitute a vast part of computer analysis in engineering. Behaviour of structures, mechanical devices, biological organs and other objects under various kinds of loads is a subject of investigation of engineers. Numerical simulation with the Finite Element Method is a major tool in these investigations. There are a few steps leading to constructing a numerical model of the problem, including identification of the geometry, material properties, support and load conditions and discretization. Responsible investigators are aware of limitations in accuracy of steps of building the model. The discretization step involved attention of researchers from the beginning of the numerical simulations era though it attracted a systematic research in the end of ’70 due to works of I. Babuska, O.C. Zienkiewicz, J.T. Oden and others. It seems that nowadays we are able to control the discretization error for a wide class of problems in computational mechanics. The purpose of this paper is to present the selected techniques of error control for problems of linear and non-linear elasticity.

2. Material models

In the simplest approach one assumes small displacements and deformation of the elastic body. In such circumstances we can take into account a linear relation between the infinitesimal strain tensor $\varepsilon(u)$ and the Cauchy stress tensor $\sigma$:

$$\sigma = C \varepsilon$$

where $C$ denotes a tensor of elasticities. In the case of finite deformations most often a model of hyperelastic material is considered which involves the density of strain energy function $\psi$ depending on the Cauchy-Green deformation tensor $C$ and its unimodular version $\tilde{C} = J^{-2/3}C$ with $J = \sqrt{\det C}$ being the volume ratio. Since the majority of elastic materials is nearly incompressible one often uses an additive split of $\psi$ into parts corresponding to $J$ and $\tilde{C}$ [6]:

$$\psi(J, C) = \psi_{v0}(J) + \psi_{iso}(C),$$

from which the constitutive relation for the second Piola-Kirchhoff stress tensor $S$ is obtained as

$$S = 2 \frac{\partial \psi}{\partial C} = -pJ C^{-1} + 2J^{-2/3} \text{Dev} \left[ \frac{\partial \psi_{iso}}{\partial C} \right],$$

with $p(J) = -\psi_{v0}(J)$ being the hydrostatic pressure.

Strong anisotropy of elastic material may be caused by the presence of reinforcing fibres along material directions $M$. In such a case the isochoric part of the strain energy function is augmented by a component depending on the auxiliary invariant associated with the structural tensor $A = M \otimes M$, namely $I_4 = C : A$, for example in the form [7]:

$$\psi_f = k_1 k_2 (e^{k_2 (I_4 - 1)^2} - 1),$$

where $k_1$ and $k_2$ are the material constants. The fibre reinforcement causes strong stiffening of the material when it is stretched along $M$ which is a similar phenomenon as near incompressibility. Analogously as for the nearly incompressible material for which we extract the volume factor $J$ depending energy we may identify the parts of $\psi_{iso}$ depending on the stretch along $M$ and on the “remaining” deformation. It allows us to use a stable numerical approximation of a nearly incompressible and nearly inextensible material with a mixed formulation with pressure and the tension associated with stretch along $M$ as separate variables [8].

3. Finite Element Method approximation

For the simplest case of linear elasticity we can state the variational formulation of the problem as the principle of virtual displacements as follows: find displacements $u \in u_0 + V$ such that

$$B(u, v) = L(v) \; \forall v \in V,$$

where $V = \{ v \in H^1(\Omega) : v = 0 \; \text{on} \; \Gamma_D \}$, and

$$B(u, v) = \int_{\Omega} \epsilon(u) : C : \epsilon(v) dx,$n

$$L(v) = \int_{\Omega} f \cdot v dx + \int_{\Gamma_N} \mathbf{t} \cdot v ds.$$
In (6) \( \Omega \) is a domain occupied by the body, \( \Gamma_N \) is the part of \( \partial \Omega \) with a Neumann boundary condition \( \sigma n = \bar{t} \), \( \Gamma_D \) the part with the Dirichlet restriction \( u = u_0 \) and \( f \) denotes body forces.

In the case of finite elasticity, as we mentioned before, due to near incompressibility (and possibly near inextensibility) we construct a mixed formulation which involves the principle of virtual displacements and, in addition, weak formulation of constitutive relations for the pressure (and possibly the tension along \( M \)). This leads to a 3- or 5-field Hu-Washizu mixed formulation. Since it is nonlinear we solve it with an adaptively constructed Newton-Raphson scheme.

4. Error estimation

Error estimation techniques have been originally developed for linear elliptic boundary-value problems and then extended to more complex approaches including nonlinearity and mixed formulations. In this generalization approximation of nonlinear problem by its linear part expressed by appropriate Gateaux derivative is exploited.

Let us consider a FE space of shape functions \( V_h \subset V \) in which we seek the approximate solution: \( u_h \in u_0 + V_h \):

\[
B(u_h, v_h) = L(v_h), \quad \forall v_h \in V_h.
\]

The most popular error estimation techniques are the residual methods. They express the measure of the error \( \| u - u_h \| \) in the energy norm \( \| u - u_h \| = \| B(u - u_h, u - u_h) \|^{1/2} \) by the residual of the FE solution \( u_h \). The most reliable methods are the sub-domain residual technique of Babuska and Rheinboldt [2], the residual method on elements proposed by Bank and Weiser [3] and the method with self-equilibrated residuals due to Ainsworth and Oden [1]. The last one is in some way superior as it expresses the error without "generic" constants:

\[
\| u - u_h \|_E = \sum_K \| \phi_K \|_E^2,
\]

where \( \phi_K \) are element error indicator functions.

A possibility of estimating the error in the energy norm allows one to estimate accuracy of selected functions defined on the solution i.e. the so-called quantities of interest (q.o.i.) which was first suggested by Becker and Rannacher [4]. In the simplest case when q.o.i. is a linear functional \( F(u) \) its accuracy is found by evaluating first the generalized Green’s function of \( F \) defined as follows:

\[
G \in V : B(G, v) = F(v), \quad \forall v \in V.
\]

The estimate reads:

\[
\| F(u) - F(u_h) \| = \| F(u - u_h) \| = \| B(G, u - u_h) \| = B(G - G_{u_h}, u - u_h) \leq \sum_K \eta_{K}^2 \eta_{K}^2
\]

where \( \eta_K^2 = \| G - G_{u_h} \|_{E,K} \) and \( \eta_K^2 = \| u - u_h \|_{E,K} \) are the element error indicators for \( G \) and \( u \), respectively.

5. Adaptivity

Reduction of the error below the prescribed level requires increasing density of discretization. This can be done either by reduction of the element sizes \( h \) or by increasing the approximation orders \( p \). A uniform modification of the mesh is prohibitively expensive. For this reason it is done locally in the areas of large error. This approach is called \( h \) or \( p \) adaptivity of the mesh. It is performed in a feed-back fashion: one solves the problem on an adapted mesh, estimates the error, adapts the mesh, and so on until the error drops below a given threshold. The most sophisticated \( hp \) version of adaptivity involves a combination of \( h \) and \( p \) refinements leading to exponential reduction of the error (w.r.t. number of degrees-of-freedom) even for singular solutions [5].

6. A numerical example

We illustrate the concept of goal-oriented adaptivity by an \( hp \) simulation of the shell-like hyperboloidal structure which is clamped at the bottom and loaded by the horizontal load \( \bar{t} = (\cos \theta, 0, 0) \) (\( \theta \) is the azimuthal angle). The height of the shell is \( h = 4 \), the semi-axes \( a = 1 \), \( c = 2 \), the thickness \( t = 0.05 \).

The \( hp \) adaptive procedure refined the mesh to reduce the error of the bending moment at the base of the shell. Figure 1. shows the final mesh (the colors represent orders of approximation: 3rd-green, 4th-yellow, 5th-orange, and 6th-red). The algorithm returned the value of the moment \( m = 0.118 \) with relative error 0.1%.

Figure 1: An \( hp \)-adaptive mesh

References


Physics and computations of turbulent dispersed flows: macro - consequences from micro - interactions

Alfredo Soldati
Dept. of Elec., Manag. and Mechanical Engineering, University of Udine
Val delle Scienze 208, 33100 Udine, Italia.
e-mail: soldati@uniud.it

Abstract

In this paper we will use Direct Numerical Simulations of turbulence and Lagrangian Particle Tracking to elucidate the physics of the motion of inertial particles in different turbulence instances and we will provide insight for modelling and simulating turbulent dispersed flows important in industrial, environmental and geophysical applications.

Keywords: turbulence; particles; Direct Numerical Simulation; Lagrangian Tracking

Turbulent fluids and small particles or droplets or bubbles are common to a number of key processes in energy production, product industry and environmental phenomena. In modelling these processes, the dispersed phase is usually assumed uniformly distributed. Indeed, it is not. Dispersed phases can be focused by turbulence structures and can have a time-space distribution which barely resembles prediction of simplified averaged modelling.

Preferential distribution controls the rate at which sedimentation and re-entrainment occur, reaction rates in burners or reactors and can also determine raindrop formation and, through plankton, bubble and droplet dynamics, the rate of oxygen-carbon dioxide exchange at the ocean-atmosphere interface.

In this talk, we will review a number of physical phenomena in which particle segregation in turbulence is a crucial effect describing the physics by means of Direct Numerical Simulation of turbulence.

We will elucidate concepts and modeling ideas derived from a systematic numerical study of the turbulent flow field coupled with Lagrangian tracking of particles under different modeling assumptions. We will underline the presence of the strong shear which flavors wall turbulence with a unique multiscale aspect and adds intricacy to the role of inertia, gravity and buoyancy in influencing particle motion. We will describe the role of free surface turbulence in dispersing and clustering the light particles such as plankton and the role of the distribution of dissipation in non-homogeneous turbulence to control breakage rates of brittle and ductile aggregates.

Figure 1: Vortices and inertial particles in a boundary layer. Different color for the vortices indicate clockwise or counterclockwise rotation in the streamwise direction. Blue particles are directed away from the wall; Purple particles are directed towards the wall.

Figure 2: Contour maps of the energy flux (panel a) and of the two-dimensional surface divergence (panel b) computed at the free surface for Re = 509.
Correlation between floater clusters and surface divergence of buoyant matter (e.g. phytoplankton, pollutants or nutrients). In a turbulent open water. This configuration mimics the motion of coherent structures responsible for particle sedimentation and re-entrainment. Specifically, we will give precise identification of features which barely resemble predictions of simplified two-dimensional modelling. In particular, in a three-dimensional open channel flow, surface turbulence is characterized by upscale energy transfer which controls the long term evolution of the larger scales. This can be demonstrated by associating downscale and upscale energy transfer at the surface with the trace of the velocity gradient tensor. A simulation snapshot elucidating this concept is presented in Figure 1.

However, turbulence features change according with the geometric features of the flow. Some significant environmental problems are relative to free surface turbulence. The free surface turbulence, albeit constrained onto a two-dimensional space, exhibits features which barely resemble predictions of simplified two-dimensional modelling. In particular, in a three-dimensional open channel flow, surface turbulence is characterized by upscale energy transfer which controls the long term evolution of the larger scales. This can be demonstrated by associating downscale and upscale energy transfer at the surface with the trace of the velocity gradient tensor. A simulation snapshot elucidating this concept is presented in Figure 2.

The presence of the inverse energy cascade at the free-surface is crucial in determining the pattern evolution of floaters and planktonic species. In particular it is possible to demonstrate that particle buoyancy induces clusters that evolve towards a long-term fractal distribution in a time much longer than the Lagrangian integral fluid time scale, indicating that such clusters over-live the surface turbulent structures which produced them [1].

A simulation snapshot elucidating this concept is presented in Figure 3.

Figure 3: Light particles floating on a flat shear-free surface of a turbulent open water. This configuration mimics the motion of buoyant matter (e.g. phytoplankton, pollutants or nutrients). Floaters segregate in $\nabla^2 D < 0$ regions (in blue, footprint of sub-surface downwelling) avoiding footprint of sub-surface upwellings). Particle buoyancy induces clusters that evolve towards a long-term fractal distribution in a time much longer than the Lagrangian integral fluid time scale, indicating that such clusters over-live the surface turbulent structures which produced them [4].

We will also discuss the effects of thermal stratification [6, 7] on the distribution of passive and active planktonic species and swimmers.

A final issue which will be addressed in this talk is the local shearing action induced by turbulence on the rupture of aggregates. Brittle and ductile aggregates will be examined and physics and statistical features of the rupture will be discussed [2, 3]. A simulation snapshot elucidating this concept is presented in Figure 4.

Figure 4: Rendering of brittle and ductile rupture in turbulent flow. The trajectory of two different aggregates is shown, superimposed onto the isosurface of the critical stress $\sigma > \sigma_{cr}$ required to produce brittle rupture or activate ductile rupture. The broken aggregate trespasses the $\sigma_{cr}$ isosurface at point A (potential brittle rupture) and undergoes ductile rupture at point B (where the breakage condition $E > E_{cr}$ is met). The unbroken aggregate avoids all regions where $\sigma > \sigma_{cr}$ and does not break within the time window considered in this figure. Critical stress isosurface is taken at the time of ductile rupture. Aggregate trajectories are tracked several time steps backward from this time.

References

Finite strain analyses of deformations in polymer specimens

Viggo Tvergaard

Department of Mechanical Engineering, Solid Mechanics, Technical University of Denmark
DK-2800 Kgs. Lyngby, Denmark
e-mail: viggo@mek.dtu.dk

Abstract

Analyses of the stress and strain state in test specimens or structural components made of polymer are discussed. This includes the Izod impact test, based on full 3D transient analyses. Also a long thin polymer tube under internal pressure has been studied, where instabilities develop, such as bulging or necking. An axisymmetric bulge develops on the tube followed by necking in the bulge, and neck propagation is observed in both the circumferential and the axial directions. Analyses of indentation tests have been carried out, with focus on the effect of the material parameters characterizing viscoplastic flow on the indentation response. Also, the ability of the simpler expanding spherical cavity model to reproduce the trends from the 3D finite element solutions has been assessed.

Keywords: indentation, viscoplasticity, polymer, computer simulation

1. Introduction

The analyses to be discussed here focus on determining the stress and strain state in test specimens or structural components made of polymer. When polymers are compressed or stretched to large strains, the plastic straining usually initiates at a stress peak, and subsequently the stress level decays during large straining until increased network stiffness gives very high stresses, as network locking essentially stops plastic deformation. Constitutive models for this type of material behaviour have been developed by Argon [1], Boyce et al. [2], Boyce and Arruda [3], Wu and van der Giessen [4]. Including models for the viscoelasticity effect has been discussed by Anand and Ames [5]. These material models have been implemented in the analyses to be discussed here, carried out in collaboration with A. Needleman.

2. Izod test

The Izod pendulum impact test is frequently used to measure the impact resistance of plastics. The test specimen is somewhat similar to the Charpy V-notch specimens often applied to test the brittle-ductile transition in structural steels, but in the Izod test the notched specimen stands clamped in a vertical position, and the pendulum strikes the clamped specimen near the free end. The specimen has standard length, depth and notch dimensions, but tests are carried out with different specimen widths, ranging from a width equal to the specimen depth (square specimen cross-section) to a width about a quarter of that value. The small width specimens give deformations under conditions near plane stress while the square cross-section specimens give higher constraint on plastic flow.

Full 3D numerical analyses have been carried out in [6] using material parameters that qualitatively represent a polycarbonate. Subsequently the effect of various deviations from these material parameters has been studied [7]. When focussing on the field quantities in the notch region it is not surprising that a band of large strains develops across the specimen. Regarding the mean stress, the maximum value is directly ahead of the notch, at the notch root. It is worth contrasting this with the distribution in metal plasticity, where the maximum mean stress occurs some distance into the material, directly ahead of the notch as a result of the increased stress triaxiality. But in the polymer the high stress at the notch results from the network stiffening, which prevents further straining but gives very large stresses. Thus, for the polymer the maximum strain with corresponding high stresses occurs first at the notch root when the limit stretch is reached, and then this region of high mean stress gradually spreads into the material from the notch root, as more material is affected by the network stiffening.

3. Tube under internal pressure

For a polymer tube loaded by internal pressure the expansion of the tube wall leads to necking. This has been studied, first under the idealized assumption of a plane strain analysis, where the necks cover the full length of the tube [8]. The tube is assumed to have an initial thickness imperfection with a number of slightly thinner regions around the circumference. At each of these thin points a neck starts to develop, and each of these necks grow into the neck propagation mode that is characteristic for polymers. For a metal the force transferred through a neck keeps reducing so that the localized necking typically ends as a fracture, but in a polymer neck the reduction of the force stops when network stiffening sets in and therefore the neck becomes longer and longer, as more material is pulled through the softening region to reach the limit stretch where network stiffening occurs.

A more realistic full 3D analysis of a long polymer tube has been carried out in [9]. In addition to the small initial thickness imperfection with a number of slightly thinner points around the circumference another initial imperfection specified a slightly larger radius at one end of the tube, because it is known that tubes made of a softening material will tend to develop a bulge instability. Indeed it turns out that a bulge forms at one end of the tube and subsequently necking of the tube wall starts to develop in the bulge.

For numerical convenience the full 3D analyses in [9] are carried out as finite element solutions of the dynamic principle of virtual work. When the pressure is prescribed to increase linearly with time, the rate of increase of the internal volume becomes large after that the maximum pressure for static loading has been exceeded. More interesting is the situation where the rate of volume increase is prescribed, as can be obtained in an experiment by pumping an incompressible fluid.
into the tube. This is done in the computation by controlling the pressure rate so that the rate of increase of the volume is approximately constant. Then the pressure goes through a maximum and subsequently reduces to a lower level where it stays rather constant during much increased deformations. The necks develop mainly in the bulge and show neck propagation as was also found in the plane strain study [8].

4. Indentation

Indentation is a frequently used test to measure the hardness of materials and to get some information on mechanical properties such as the yield strength and the elastic modulus. For polymers full 3D analyses of indentation have been carried out in [10], considering conical and pyramidal indenters. Also the spherical cavity model was considered, as suggested for metals in [11] and discussed in [12]. It was observed that the subsurface displacements produced by a blunt indenter are approximately radial from the point of first contact, with roughly hemispherical contours of equal strain. This model requires only the 1D analysis of the expansion of a spherical void.

The analyses in [10] use material parameters with a large limit stretch that give a reasonable approximation of tensile tests for a high-density polyethylene. In [13] the same type of computations are carried out for a number of different sets of material parameters all giving the up-down-up tensile or compressive law characteristic for polymers, some of them with a smaller limit stretch. All the analyses in [10, 13] show that also for the polymers the hardnesses predicted by the full 3D numerical analyses are rather well approximated by the spherical cavity model where the analyses are 1D with spherical symmetry.

It is found that two material parameters specifying the shape of the initial peak of the tensile or compressive curve have a strong influence on the hardness. For the same values of these two parameters it turns out that large changes in the network stiffening, as described by the limit stretch, have rather little influence on the predicted hardness.

For metals it is well known that the hardness is about three times the yield strength [12]. But if the initial peak stress on the uniaxial tensile or compressive curve is considered the effective yield strength of the polymer, the hardness is much lower than a factor three times this yield strength. The effect of elastically soft or plastically compressible solids on the hardness has recently been investigated [14]. The plastic deformations in the computations [10, 13] for polymers were taken to be incompressible, but the ratio of Young’s modulus to the effective yield strength for these materials was only 19, much lower than the value of this ratio for metals. Therefore, the hardness found for the polymers is only about 1.05 times the effective yield strength.

References


Mini-symposia
Adaptive Methods and Error Estimation

organized by W. Cecot, W. Rachowicz and G. Zboiński
The application of model parameter estimation method to detect connection damage in a steel-concrete beam using modal force residuals

Malgorzata Abramowicz
Faculty of Civil Engineering and Architecture, West Pomeranian University of Technology Szczecin
Piastów Ave. 50/309, 70-311 Szczecin, Poland
e-mail: mabramowicz@zut.edu.pl

Abstract

The main topic of the paper is damage detection of steel-concrete composite beams using estimation methods for parameter identification. The paper presents the issues of modelling spatial vibration of steel-concrete composite beams often used as main elements of composite floors or in bridge engineering as the main load-carrying girders. A discrete, spatial, computational model for a steel-concrete composite beam was developed using the Rigid Finite Element Method. RFEM allows for effective determination of dynamic properties of beams. In order to validate the algorithms and to demonstrate the usefulness of the 3D model, it was used to detect the size of damage. Analysis was focused on damage in steel connectors that join the reinforced concrete slab with the steel section. The simulation of damage detection confirmed high effectiveness of the developed algorithms.

Keywords: detect damage, rigid finite element (RFE) model, steel-concrete composite beams, modal parameters, vibrations

1. Introduction

Structure diagnosis is one of the most important elements in engineering practice. One of the commonly used methods for damage detection is modal analysis. The main topic of the paper is damage detection of steel-concrete composite beams using previously developed methods for parameter identification [1]. The algorithms of parameter identification are based on the comparison of experimental and calculated natural frequency modes. To validate the algorithm and to demonstrate the usefulness of the 3D model, it was used to detect the size of damage. Analysis was focused on damage in steel connectors that join the reinforced concrete slab with the steel section.

2. Materials and computational model

The analysed composite beams consisted of a rolled-steel I - beam IPE 160 made from S235JRG2 steel, connected to a concrete slab with a cross section of 60x600 mm manufactured from C25/30 concrete. A ductile connection was made using headed studs manufactured by KÖCO – SD type, 10 mm in diameter and 50 mm in height, made from S235J2G3 steel. The studs were placed in pairs every 200 mm. The total length of one beam was 3200 mm (Fig. 1).

![Figure 1: Composite beam C1: a) the cross-section; b) side view](image)

3. Identification algorithm for connection damage

Damage was modelled modifying the values of stiffness coefficients in spring-damping elements connecting RFEs that model the reinforced concrete slab with RFEs that model the steel section. Damage was introduced into the model by means of decreasing the percentage values of parameters that defined a single SDE, i.e. the shearing stiffness \( K_h \), the axial stiffness \( K_v \) and the rotational stiffness \( K_{R,X} \).

Damage detection process is divided into two stages. The first stage consists of the location of the damage [4]. In the second stage, stiffness parameters characterising the connection between the reinforced concrete slab and the steel section \( K_{H,DET} \), \( K_{A,DET} \) and \( K_{R,X,DET} \) were estimated. Damage was modelled in different locations along the length of the composite beam.

Damage detection was determined calculating the natural frequency modes and eigenmodes of modelled, undamaged beams, for parameter values obtained in parameter estimation that was based on experiments. The characteristics of eigenmodes were transformed into measuring points so that simulated damage detection would reflect the actual damage detection based on measuring points. Then, the model was modified and one or more pairs of headed studs was damaged. Calculations were made again and natural frequency modes and eigenmodes were determined for the modelled damage. Consequently, two sets of data were obtained: one for an undamaged composite beam and the other for a damaged beam.
4. Damage detection using modal force residuals

In order to estimate connection parameters for the damaged beam, whose values would reflect the extend of damage to steel studs, damage location was crucial. The method of residual modal forces was used to this end. The vector of residual modal forces was determined from Equation (1) [5], [6]:

\[ \mathbf{E} = \mathbf{K} \Phi \mathbf{A}', \]

where:

- \( \mathbf{E} \) – residual modal forces matrix;
- \( \mathbf{K} \) – global stiffness and mass matrices;
- \( \Phi \) – mode shape matrix of damaged structure;
- \( \mathbf{A}' \) – natural frequency diagonal matrix of damaged structure.

The residual modal forces for the undamaged structure values were equal to zero, and for the damage regions the values were nonzero. To locate damage in the studs, the method of residual modal forces was implemented into a program in MATLAB environment. Consequently, the program provided data concerning damage location in the studs. Having found damaged places, the size of damage was determined. The stiffness values \( K_v, K_h \) and \( K_{R,X} \) for localised damage in the steel studs was investigated for in the damaged model. The parameters were estimated using the determined values of natural frequency modes. Eventually, a value was found that described stiffness decrease for a given pair of steel studs.

5. Simulation verification for the steel – concrete beam

The above algorithm was used in several series of numerical experiments conducted for the composite beam C1. The computational numerical model was determined with identified parameters and random noise was not introduced into it. Figure 2a presents the composite beam C1 with numbered pairs of steel studs. Three types of damage were made for the beam (Fig. 2b).

![Figure 2](image)

Figure 2: Schematic of the composite beam C1 with selected pairs studs: a) spatial view; b) side view of the marked damage pairs studs

In the first damage, DET_1, the connection between a stud and concrete was weakened by decreasing the stiffness parameters \( K_v, K_h \) and \( K_{R,X} \) by 30%. DET_1 was modelled at the beginning of the composite beam. The second damage, DET_2, modelled a 90% connection loss between a stud and concrete, resulting in the stiffness parameters \( K_v, K_h \) and \( K_{R,X} \) decreased by 90%. DET_2 was modelled in the middle of the beam. In the third damage, DET_3, damage of two pairs of steel studs was modelled. 90% for one pair and 30% for the other. DET_3 was modelled in the middle and at the end of the beam. The coefficient \( \Delta_{DET} \) stands for the percentage value of damage in the connection, i.e. the magnitude of decrease for the stiffness parameters \( K_v, K_h \) and \( K_{R,X} \).

6. Results

In numerical simulations calculations were made first for undamaged and then for damaged models. The results were used to localise damage. Finally, the stiffness parameters for damaged connections \( K_{v,DET}, K_{h,DET} \) and \( K_{R,X,DET} \) were estimated. Simulation results for all types of damage for the beam C1 are shown in Table 1. The coefficient \( \Delta_{DET} \) stands for the percentage value of damage, based on the estimation of damaged parameters. The damage detection simulations confirmed high effectiveness of the developed algorithms.

| Table 1: Simulation results for all types of damage |
|-----------------|---------|-------|-------|
| **BEAM C1**     | DET 1  | DET 2 | DET 3 |
| No damaged pair of steel studs | 3      | 10    | 7     |
| The percentage value of damage \( \Delta_{DET} \) [%] | 27     | 90    | 94    |

Damage location data was consistent and the obtained damage size data was close to expectations. The developed method of model parameter estimation may provide an effective tool for damage detection. Further research will focus on testing the method on actual steel-concrete composite beams.

References


Study of convergence of the multigrid homogenization

Witold Cecot¹, Marta Oleksy², Michał Krówczyński¹*

¹,²,³ Civil Engineering Department, Cracow University of Technology
Warszawska 24, 31-155 Cracow, Poland
e-mail: plcecot@cyf-kr.edu.pl¹, moleksy@L5.pk.edu.pl², m.krowczynski@L5.pk.edu.pl³

Abstract

The FEM approximation of problems with heterogeneous materials often leads to a huge number of degrees of freedom. One of the promising methods for such a reduction of the number of unknowns that preserves the most important fine mesh data is the multigrid technique. This algebraic homogenization proved to be an efficient method of upscaled solution approximation for problems with rapidly varying, possibly noncontinuous coefficients as well as with non coherent (porous) domains that may or may not depict a periodic micro structure. Our main contribution is an improved, appropriate for bubble functions, intergrid mapping and consequently a fast convergence of algebraic error for both displacements and stresses that was observed in the conducted numerical experiments.

Keywords: heterogeneous materials, algebraic homogenization, error convergence

1. Introduction

In the last years the most intensively used method for analysis of multiscale problems is the computational homogenization. In our research we develop the multigrid homogenization, which may be used for both periodic and non-periodic materials. The higher order FEM approximation at the macro-scale and a new definition of the intergrid operators leading to a fast convergence of both displacements and stresses was proposed in [4]. In this paper, for the sake of simplicity, only two meshes are considered, fine one that resolves locally the highly oscillating material properties and a coarse mesh for global, low cost computation. Thus, such an approach results in evaluation of both the mean global field and the local fluctuations of displacements as well as stresses.

Generally, the multigrid method [3] may be used in two different ways for heterogeneous materials. Either, in a special version accommodated for fast varying material parameters in order to obtain efficiently a direct numerical solution on the finest grid [1] or as an upsampling method [6] leading to algebraically homogenized solution on the coarsest mesh. The latter method is equivalent to the Multiscale FEM (MsFEM) [5], in which special shape functions are constructed, similarly as it was proposed in [2], to resolve all the details of material heterogeneities. Typically first order shape functions are used. Only recently [7] higher order Lagrange type bases were used without thorough convergence study.

2. Formulation

We present in this section the basic idea of the multigrid homogenization after [4]. Let’s consider the well known linear elasticity problem in 2D with heterogeneous material: find field of displacements \( u(\mathbf{x}) \) such that:

\[
-\frac{\partial}{\partial x_1} \left( C_{ijkl} \frac{\partial u_k}{\partial x_i} \right) = f_i \quad \forall \mathbf{\omega}_k \subset \Omega \quad (1)
\]

with Dirichlet (\( \hat{u} \)) and Neumann (\( \delta \)) boundary conditions on \( \partial \Omega_D \) and \( \partial \Omega_N \) respectively as well as the continuity conditions at the material interfaces between \( \mathbf{\omega}_k \) subdomains with continuous material parameters.

Let the FEM system of algebraic equations be written in the following matrix form

\[
K^h \mathbf{u}^h = f^h \quad (2)
\]

where \( \mathbf{u}^h \) is the vector of dof and \( K^h, f^h \) denote the assembled matrix and vector.

Since solution of (2) may be computationally too expensive one may approximate it by a coarse mesh solution \( (\mathbf{u}^H) \) defined by the following linear equations

\[
K^H \mathbf{u}^H = f^H \quad (3)
\]

However, the system (3) and consequently the dof vector \( (\mathbf{u}^H) \) must account, at least implicitly, for the material heterogeneity, e.g. by the multigrid variational coarsening, i.e. by computing

\[
K^H = I^T K^I I, \quad f^H = I^T f^h \quad (4)
\]

where \( I \) denotes interpolation operator that transfers the coarse mesh degrees of freedom (dof) into the fine mesh dof. The \( I \) operator is the key component of the multigrid homogenization and is constructed locally for every coarse mesh element.

We proposed [4] to compute the \( I \) operator as the solution of the following boundary value problem defined for every coarse mesh element \( L_H \): given \( \Psi \), find \( \Phi \) such that

\[
\frac{\partial}{\partial x_1} C_{ijkl} \frac{\partial \Phi_k}{\partial x_i} = Reg \left( \frac{\partial}{\partial x_1} C_{ijkl} \frac{\partial \Psi_k}{\partial x_i} \right) \quad \forall i = 1, 2, \mathbf{x} \in L_H \quad \Phi = \hat{\Phi} \text{ on } \partial L_H \quad (5)
\]

where \( Reg \) denotes the regular part of the derivative, i.e. without distributional part and \( \hat{\Phi} \) consists of scalar valued functions obtained for every nonzero trace of all coarse element scalar

*This research was supported by The National Science Center under grant 2011/01/B/ST6/07312.
shape functions $\psi$ as the solution to the following 1D boundary value problem
\[
\frac{d}{ds} (2\mu + \lambda) \frac{d\hat{\varphi}}{ds} = \text{Reg} \left[ \frac{d}{ds} (2\mu + \lambda) \psi \right] \quad \forall s \in (0, l)
\]
\[
\hat{\varphi}(0) = \psi(0), \quad \hat{\varphi}(l) = \psi(l)
\] (6)

3. Numerical examples

Although the multigrid homogenization does not make any assumption on periodicity it is convenient to study its accuracy and convergence for a periodic material. In the next test we assumed a rectangular domain with a periodically distributed circular holes of radii $r=0.1412$ (Fig. 1) in a plane strain state. Material parameters were $\nu=0.1$ and $E=180$. A constant, distributed loading of value 1 was applied at the right edge.

A fine discretization with about 47000 dof was used as a substitute for the exact solution. Our experimental study shows even exponential convergence (see Figs. 2, 3) both in $L_2$ and energy norms for higher order approximation in the coarse scale. The error (called an algebraic one) is defined as
\[
e = u^h - Iu^H
\] (7)

The coarse mesh consisted of either 1x1 or 2x2 or 4x4 square elements with integrated Legendre polynomials of order 1 to 5 as the shape functions.

4. Concluding remarks

We applied the hierarchical approximation of order up to 5 for multigrid homogenization and a new method of the intergrid mapping construction. Whenever these mappings are well defined the coarse element stiffness matrices and load vectors are computed by multiplication of previously evaluated matrices and vectors without necessity for additional integration. The numerical experiments show a fast reduction of the algebraic, which is an additional modeling error that inevitably accompanies any homogenization. The fast convergence is observed for both displacements and stresses while higher order of coarse scale bases are used. We also pointed out that the multigrid homogenization is equivalent to the MsFEM. Further development of this type of homogenization will include application to physically and geometrically nonlinear problems as well as more than two grid coarsening.

References

Integration of \textit{hp}-adaptive FEM and local numerical homogenization

Marek Klimczak\textsuperscript{1}, Witold Cecot\textsuperscript{2}\thanks{This research was sponsored by the Polish National Science Center (NCN) under project UMO-2011/01/B/ST6/07312.}

\textsuperscript{1,2}Faculty of Civil Engineering, Cracow University of Technology
Warszawska 24, 31-155 Cracow, Poland

\textsuperscript{1}e-mail: mklimeczak@L5.pk.edu.pl, \textsuperscript{2}pleceot@cyf-kr.edu.pl

Abstract

The paper presents a novel approach to modeling heterogeneous materials by means of two techniques, i.e. well established \textit{hp}-adaptive FEM and local numerical homogenization (LNH). These methods are naturally compatible. They make it possible to use optimally the computer power without losing the accuracy of the solution. A brief description of the applied methods is given in the paper. Subsequently, the idea of their integration is described and illustrated by a numerical test to assess accuracy of such a homogenization.

\textit{Keywords:} local numerical homogenization, adaptivity

1. Introduction

Even though the computational power of todays computers is increasing rapidly, it is still limited. In context of heterogeneous materials it means that there is a scale of analysis, not to be directly taken into account due to the computational cost. Thus, a variety of homogenization methods is used to enhance the modeling process. In the research an interesting technique of local numerical homogenization is used. This approach is integrated with the \textit{hp}-adaptive FEM.

2. Local numerical homogenization

A detailed description of the method can be found in [2, 3]. Our contribution, i.e. modification of the method towards nonlinear problems analysis, was presented in [4, 5]. A general algorithm is as follows:

- generate the mesh as fine as possible (e.g. due to computer power and time limitations), which henceforth will be called a coarse one,
- refine its elements in order to match the inhomogeneity (refinement within neighboring elements is not required to be compatible),
- compute the effective stiffness matrices for the initial (coarse) mesh elements,
- solve the problem using the coarse mesh.

For the sake of clarity both the above algorithm and further description of the integration of the \textit{hp}-adaptive FEM and local numerical homogenization are presented only with reference to linear elastic problems.

The core of the entire approach is the computation of the effective stiffness matrix. It is based on the minimization of the norm of the difference between the coarse and fine mesh would-be solutions. Regularization is also necessary in order to obtain a unique solution. Formulation of the problem is presented below.

For a non-zero load vector \( f \), known symmetric stiffness matrices for fine mesh elements \( K \), interpolation matrix \( A \), positive-definite symmetric matrix \( B \), dimensionless small parameter \( \epsilon > 0 \), a symmetric effective matrix \( \hat{K} \) is investigated minimize \( E \), where:

\[
E(\hat{K}) = \frac{1}{2} \| (\hat{K} - A\hat{K}^1A^T) f \|_B^2 + \frac{\epsilon}{2} \| K - A\hat{K}^1A^T \|_F^2 \| f \|_B^2 \tag{1}
\]

A fine mesh load vector is determined on the basis of the auxiliary solution. On the basis of the element solution the strain and the stress tensors are computed at Gauss points on the faces. Then the tractions are evaluated and equilibrated if required. The main advantage of the local numerical homogenization is that there is no restriction on the separation of analysis scales. This condition may exclude different (e.g. RVE-based) computational homogenization methods in some practical engineering problems.

A schematic idea of the proposed approach is shown in Fig. 1. Although the product of the LNH is just the effective stiffness matrix, also the effective material properties can be computed.

![Figure 1: Scheme of local numerical homogenization (for a single coarse element)](image)

3. An \textit{hp}-adaptive FEM with local numerical homogenization

There are three main methods of mesh adaptivity:

- \( h \) refinement, which is based on the reduction of element size,
- \( p \) refinement - based on increasing of the approximation order and
4. Numerical results

In order to illustrate the developed approach a simple uniaxial tensile test is performed on a cuboid specimen. The load is distributed uniformly ($q = 100$) on the left hand face. The three perpendicular faces (right, bottom, front) are subjected to no-penetration boundary conditions. We investigate the error introduced by local numerical homogenization analyzing the heterogeneous material shown in Fig. 2. The length of the specimen is equal to 2, its cross-sectional dimensions are 1×1. A single inclusion is a cube of the edge length equal to $\frac{1}{4}$. We analyzed linear elastic behavior of the material ($E = 2000\varepsilon/6$ for the matrix, $E = 200\varepsilon/6$ for the inclusions, $\nu = 0.3$). First we obtained a reference solution of the problem on a uniformly refined mesh. Then the problem was solved using local numerical homogenization (with only two coarse mesh elements). The maximum values of horizontal displacements were compared. A relative error introduced by local numerical homogenization can be expressed as:

$$\frac{\|u_{h}^{ref} - u_{h}^{num}\|}{\|u_{h}^{ref}\|} = \frac{1.996e^{-4} - 1.057e^{-4}}{1.996e^{-4}} = 0.02.$$

The plot of horizontal displacements is presented below in Fig. 3. In the case of both fine and coarse discretization linear shape functions were used. Thus, the total number of coarse mesh DOFs is equal to 36. In the case of a refined mesh this number is equal to about 2000 for both coarse elements. In our research we take advantage of a well-established version of automatic hp-adaptive FEM described in detail in [1]. Starting with an initial (relatively coarse) mesh we end up with an optimal mesh, which minimizes projection-based interpolation error at a minimum computational cost. The initial mesh can be generally refined in an anisotropic way.

A hp-adaptive FEM serves in the routine presented in Chapter 2 twice. It is used first as a coarse mesh ‘generator’ at the first step of the algorithm. Even though the mesh is called ‘coarse’, in fact it is the optimal mesh we can afford. This auxiliary solution is used in the next step to state boundary conditions for a single coarse element. For the analysis of this subdomain the hp-adaptive FEM is also used. On the basis of assumed interpolation matrix mapping coarse and fine DOFs, assembled fine mesh load vector and stiffness matrices the effective stiffness matrix for a coarse element may be obtained. Subsequently, the main problem can be solved reading in stiffness matrices computed in the previous step.

5. Final remarks

In the paper a brief description of local numerical homogenization and its integration with hp-adaptive FEM was presented. A preliminary numerical example was also provided in order to illustrate the application of the proposed approach. Convergence of the method will be numerically examined and presented during the conference. Considerable reduction of the DOFs number can be observed with no disruption on the solution accuracy. Analyses of coarse elements leading to assignment of their effective matrices are independent. Thus, preliminary tests were performed on parallelization of the routine. The results are promising. Development of this approach is desired in context of real structural transient analysis.

Further research effort bounds to propose reliable error estimates in order to measure an introduced homogenization error. Application of the method to nonlinear problems will be continued too.

References


The influence of different equivalent boundary conditions on approximate solution to a potential problem

Jan Kucwaj

Institute for Informatics, Department of Physics, Mathematics and Informatics, Cracow University of Technology, ul. Warszawska 24, 31-155 Cracow, Poland,
e-mail: jkucwaj@pk.edu.pl,

Abstract

The main goal of the paper is to compare numerical approaches obtained by stating boundary conditions on different connected components of the boundary. The problem considered is a potential flow around a profile [2, 6]. It can be shown that in case of a rectangular computational domain with two sides perpendicular to the speed direction the potential function is constant on the connected components of these sides. It allows to state the Dirichlet conditions on the considered parts instead of the potential jump on the slice connecting the trail edge with the external boundary. The adaptive remeshing method [2, 3, 4] to the solution of the considered problem was applied.

Keywords: adaptation, remeshing algorithm, Delaunay triangulation, grid generation, potential flow, Kutta-Joukovsky condition, nonlinear problem.

1. Introduction

The main goal of the paper is to compare approximate solution to a potential flow problem by stating formally different but physically equivalent mathematical formulations. The difference lies in stating Dirichlet conditions on different connected components of the perpendicular part of the boundary of the rectangular computational domain to the speed direction. Assuming that the speed in the infinity has horizontal direction, the vertical component of speed vanishes, it means that the speed is constant on the vertical component of the boundary. The value of the speed is chosen to that satisfy Kutta-Joukovsky condition. The choice of the connected component acts on the speed of the convergence of the secant method used to solve the nonlinear algebraic equation representing Kutta-Joukovsky condition. It can be mentioned that the loop over secant method is external to the loop of the solution of nonlinear system of algebraic equations representing discretized form of the physical problem. The whole problem is led to the solution a system of two equations i.e. a nonlinear elliptic equation of the second order and nonlinear algebraic equation representing Kutta-Joukovsky condition. It can be mentioned that the loop over secant method is external to the loop of the solution of nonlinear system of algebraic equations representing discretized form of the physical problem. The whole problem is led to the solution a system of two equations i.e. a nonlinear elliptic equation of the second order and nonlinear algebraic equation representing Kutta-Joukovsky condition. The parameters defining the coefficients of the the Kutta-Joukovsky condition depend on the solution of the differential equation. The algebraic system of equations is solved for every step of iteration of the secant method used in the solution of Kutta-Joukovsky condition. An adaptive method based on numerical grid generator with a mesh size function [2, 3, 4] is applied.

2. Problem formulation

It is assumed that the considered flow is stationary, irrotational, compressible and inviscid in domain $\Omega$ around the profile $P$ (Fig. 1). The following notations are used:

- $\Gamma_P$ - the boundary of the obstacle $P$,
- $\Sigma$ - the slit from $A$ to $B$,
- $\Gamma_\infty$ - external part of the boundary $\Omega$.

The boundary of $\Omega$ has the following connected parts:

$$\partial \Omega = \Gamma_\infty \cup \Gamma_P.$$  (1)

The boundary value problem of steady flow is stated as follows:

$$\nabla \cdot [\rho (\nabla u^2)] = 0 \quad \text{in} \quad \Omega,$$  (4)

$$\frac{\partial u^+}{\partial n} - \frac{\partial u^-}{\partial n} = 0 \quad \text{on} \quad \Sigma \quad \text{where}$$  (5)

$$u^+ \text{ and } u^- \text{ are values of } u \text{ over upper}$$  (6)

and lower part on the slit, respectively, and

$$u^+ - u^- = \beta \quad \text{for some jump } \beta \text{ on} \quad \Sigma,$$  (8)

$$\rho (|\nabla u|^2) \frac{\partial u}{\partial n} = \rho (|u_\infty|^2) v_\infty n_\infty \quad \text{on} \quad \Gamma_\infty,$$  (9)

where $n_\infty$ is the external normal to $\Gamma_\infty$, and

$$\rho_\infty = \rho (|v_\infty|).$$  (11)

In order to determine the jump $\beta$, we need some additional Kutta-Joukovsky condition

$$K(\beta) = |\nabla u^+|^2 - |\nabla u^-|^2 = 0 \quad \text{at the trail edge}.$$  (14)

where $u$ is the speed potential $v = \nabla u$.

and $\rho (|\nabla u|^2) = \rho_0 (1 - \frac{\kappa - 1}{2a_0^2} |\nabla u|^2)^{\frac{1}{\kappa - 1}}$  (15)

is the gas density. Here $\kappa > 1$ is the adiabatic gas constant, e.g. $\kappa = 1.4$ for dry air. The constant $\rho_0, a_0$ are the density and the local speed of sound, respectively, for the motionless gas.
3. Alternative equivalent formulation of the problem

It is possible to formulate three equivalent problem of the potential flow 4. All of them differ from 4 by replacing the boundary condition 8 by:

a) \( u = \beta \) on \( \Gamma_1 \), (Fig.1) 

b) \( u = \beta \) on \( \Gamma_2 \), (Fig.1) 

c) \( u = \beta \) on \( \Gamma_3 \), (Fig.1)

4. The algorithm of remeshing based on grid generator

The algorithm of remeshing can be divided into the following steps:

1. preparation of the information about the geometry and boundary conditions of the problem,
2. fixing an initial mesh size function,
3. mesh generation with the mesh size function,
4. solution to the problem 4 on the previously generated mesh,
5. evaluation of error indicator in every element,
6. calculation of nodal error indicator values by using average method,
7. definition of the new mesh size function using the errors computed at every point,
8. if error not satisfactory go to point 3.,
9. end of computations.

The applied indicators are calculated for every element or directly at the nodes [2, 4]:

Let \( e_i \) for \( i = 1, \ldots, n_0 \) be an error indicator at \( i \)-th apex of the mesh \( T_0 \), and \( P_0=\{ P_i, i=1, \ldots, n_P \} \) - set of nodes. We define a patch of elements for every node \( P_i \) as:

\[ L_i = \{ k : P_i \in T_k \} \text{ for } i = 1, \ldots, n_P. \] (19)

The modification of the mesh size function is performed at every adaptation step to perform the upcoming one. The main idea of this part of the algorithm relies on reduction of the values of the mesh size function by an appropriately chosen function. The chosen function is continuous, linear and its smallest value is at the node of the maximum error indicator and the greatest value where the error is minimum. The function increases due to the error decrement.

5. Numerical experiments

Numerical experiments prove that the method is most efficient in case of the stating Dirichlet conditions for boundary \( \Gamma_3 \). It can be proved that there exists a unique solution for each of these three equivalent formulations. In all cases all physical conditions are satisfied.

References

Stress convergence in adaptive resolution of boundary layers in the case of 3D-based first-and higher-order shell models

Łukasz Miazio¹, Grzegorz Zboiński¹,²∗

¹Faculty of Technical Sciences, University of Warmia and Mazury
ul. Oczapowskiego 11, 10-719 Olsztyn, Poland
e-mail: lukasz.miazio@uwm.edu.pl

²Institute of Fluid Flow Machinery, Polish Academy of Sciences
ul. Fiszera 14, 80-231 Gdańsk, Poland
e-mail: zboi@imp.gda.pl

Abstract

The presented paper concerns stress convergence analysis in the case of the boundary layers in thin-walled structures modeled with the 3D-based first-order Reissner-Mindlin model and the 3D-based higher-order shell models as well. We consider both parametric and adaptive convergence of various boundary stress components. The main objective of this research is to demonstrate that the appearance of elongated elements in the vicinity of the boundary does not destroy numerical stability of the solution of the problem. Such elements are generated in the second, modification step of the four-step adaptive strategy being our extension of the so-called Texas three-step strategy. The appearance of the elongated elements is a result of application of numerical tools for a posteriori detection and significance, intensity and division ratio assessments of boundary layers, followed by the corresponding mesh modification procedures.

Keywords: stress convergence, boundary layers, adaptive resolution, a posteriori detection

1. Introduction

In the research is investigated local convergence properties of the boundary stresses in the case of the edge effect occurring in thick- or thin-walled elastic structures described with 3D-based shell models. The 2D, classical and hierarchical shell models were analysed in this context in [7] and [8], respectively. Global convergence of the solution of the 3D-based models was investigated by the authors in [2] and [9] in the strain energy norm.

2. Detection and resolution of boundary layers

2.1. The detection and assessment tools

In order to detect the boundary layer phenomenon a kind of sensitivity analysis is performed. The analysis is based on comparison of two local solutions obtained from the local meshes suitable for the cases of existence and lack of the phenomenon. The local problems are generated through the division of a couple of prismatic elements adjacent to the boundary either exponentially or arithmetically into four smaller elements (Fig. 1).

Figure 1: Meshes of the local problems [2]: a) before the division, b) arithmetic division into halves, c) exponential division

In the case of detection and significance assessment of the phenomenon one solution for the standard exponential mesh is taken into account only, while in the case of intensity and optimal division ratio assessments the proper sequences of the exponential meshes are considered [2,9]

2.2. Our adaptive modification of the mesh

The modification (second) step of the four-step adaptive procedure consists in exponential subdivisions of the layers of elements adjacent to the boundaries affected by the edge effect. Each such a boundary can be modified independently. Additionally, the presence, the number of subdivisions and the exponential subdivision ratio may vary for each boundary. The details of the mesh modification procedures can be found in [2]. The proposed modification step completes standard three-step hp-adaptive algorithm, called a Texas three-step strategy and presented in [7].

3. Numerical tests

Numerical experiments concern the model problem of a square plate. Because of symmetry of the plate geometry, loading and boundary conditions only a quarter of the plate is analyzed. The plate length is \(l = 1.57075\times10^{-2} \text{ [m]}\), while its thickness is equal to \(t = 0.03\times10^{-2} \text{ [m]}\). The Young’s modulus is \(E = 2.11\times10^{11} \text{ [N/m²]}\), the Poisson’s ratio equals 0.3. The plate is loaded with the uniform normal traction \(p = -4.0\times10^{6} \text{ [N/m²]}\) and clamped around its lateral boundaries. We apply first-order and hierarchical shell models of the transverse approximation orders \(q\) equal to 1 and 2, respectively.

3.1. Parametric studies

In the presented tests we apply a starting, regular, 3x3 mesh shown in Fig. 2. The respective numbers of the finite elements of this mesh are presented in the figure.

*This work was partly supported by the research project no. 5153/T02/B/2011/40 of the National Science Centre.
Figure 2: The global numbering of elements (3x3 regular mesh)

In the tests concerning local convergence of the boundary stress components the starting mesh is modified by dividing it exponentially towards outer boundaries of the plate. Such modification is performed in the layers of elements adjacent to two outer edges of the plate. The division concerns all elements in these layers. A dimensionless division ratio $\rho$ describes the size ratio of the elements obtained by means of modification. The divisions are shown in Fig. 1. In the case of $\rho = 1$ the division into halves is done, while $\rho = 10$ and $\rho = 100$ correspond to the exponential divisions. For each of three cases a $p$-convergence analysis is performed, where $p$ stands for the longitudinal order of approximation. The values of this order were set to 4, 6 and 8. We checked how all six stress components, and the effective stress, change at the centroid of the element lateral face adjacent to the outer boundary of the plate. The elements considered were numbered 8 and 16 (Fig. 2). The presented results concern a hierarchical shell model, presented in Figs. 3 and 4 for the cases of the effective $\sigma_{ef}$ and normal $\sigma_{nn}$ stresses.

Figure 3: Effective stress convergence at the outer boundary

3.2. Adaptive analysis

Additionally, a $hp$-adaptive analysis was performed of the same model plate problem, to check the convergence of stresses in the case of a four-step strategy. The strategy included exponential modification of the mesh as a second step, in the vicinity of the boundaries. The obtained results confirmed stable convergence properties of each individual stress component and the effective stress, too.

Figure 4: Normal stress convergence at the outer boundary

4. Conclusions

The results obtained in convergence studies of stresses on the boundaries affected by the edge effect have confirmed the anticipated numerical stability of the solution.

The presence of the elongated elements in the vicinity of the boundaries does not destroy the solution convergence both in the global sense (in terms of the strain energy of the plate) and in a local, boundary sense (in terms of the boundary stress values). Our results stand in good agreement with the analogous results obtained by other researchers in the case of the 2D, both classical and hierarchical shell models.

References


Computable double-sided a posteriori error estimates for \( h \)-adaptive Finite Element Method

Oleksandr Ostopov\(^1\), Oleksandr Vovk\(^2\), Heorhiy Shynkarenko\(^3\)

\(^1,2,3\)Department of Information Systems, Ivan Franko National University of Lviv
Unversytetska 1, 79000 Lviv, Ukraine
e-mail: oleksandroatopov@gmail.com\(^1\), olexandrovovk@gmail.com\(^2\), shynkarenko@po.opole.pl\(^3\)

\(^3\)Department of Mathematics and Applied Informatics, Opole University of Technology
Luboszycka 5, 45-036 Opole, Poland
e-mail: h.shynkarenko@po.opole.pl

Abstract

A simple element-wise a posteriori error estimators (AEEs) is described providing double-sided error estimates for finite element method (FEM) approximations of the solutions to the elliptic boundary value problems. We name these estimators "Dirichlet and Neumann estimators" because the theoretical background of their double-sided estimations lays in the analysis of the solutions to the variational formulations of the residual problems with homogeneous Dirichlet and Neumann boundary conditions. It is shown how to construct Dirichlet and Neumann AEEs for the linear finite element approximations on the triangular meshes. Numerical results demonstrate the properties of Dirichlet and Neumann AEEs in the process of solving the singularly perturbed and non-linear diffusion-advection-reaction problems on the uniform and \( h \)-adaptively refined meshes.

Keywords: double-sided a posteriori error estimates, element-wise Dirichlet and Neumann a posteriori error estimators, linear and bilinear finite element approximations, \( h \)-adaptive mesh refinement

1. Introduction

A posteriori error estimators have already become a standard supplement to the finite element method [1], [2], [3]. Here we continue the research [5], [6], [7] constructing Dirichlet and Neumann AEEs for the double-sided error estimation of FEM approximations. The ability to compute the element-wise approximation of the lower and upper FEM error bounds expands the control and correction options of the boundary value problems solving with the aim to increase its efficiency. In the paper correction consists in \( h \)-adaptivity widely used for solving singularly perturbed problems, see [4]. Control means here the possibility of error approximation refinement is especially important.

2. Problem formulation

We consider an elliptic boundary value problem with a weak non-linearity in the right-hand side:

\[
\begin{align*}
-\nabla \cdot (\mu \nabla u) + \beta \cdot \nabla u + \sigma u &= f(u) \text{ in } \Omega \\
\mu \nabla u &= 0 \text{ on } \Gamma_D, \\
-\beta \cdot \nabla u &= g \text{ on } \Gamma_N = \partial \Omega \setminus \Gamma_D.
\end{align*}
\]

(1)

that has the following variational formulation:

\[
\begin{align*}
\text{find } u \in V := \{ v \in H^1(\Omega) : v = 0 \text{ on } \Gamma_D \} \text{ such that } \\
a_\Omega(u, v) &= n_\Omega(u; v) \quad \forall v \in V, \text{ where } \\
a_\Omega(u, v) &= \int_\Omega [\mu \nabla u \cdot \nabla v + \nu \beta \cdot \nabla u + \sigma uv] \, dx, \\
n_\Omega(u; v) &= \int_\Omega f[u]v \, dx - \int_{\Gamma_N} g v \, d\gamma, \quad \forall u, v \in V.
\end{align*}
\]

Here we assume that the given functions \( \mu = \mu(x), \beta = \beta_i(x), \sigma = \sigma(x), f[u] = f(x, u) \) are such that the problem (1) has a unique solution \( u = u(x) \).

Below we suppose that Newton method was used for the linearization of the problem (1) and FEM was used to obtain the approximation \( u_h = u_h(x) \) of the solution \( u = u(x) \) in the finite dimensional subspace \( V_h \subset V, \dim V_h = N(h) < +\infty \).

3. A posteriori error estimators

In order to estimate the quality of the approximation \( u_h \) obtained on the partition \( \mathcal{T}_h = \{ K \} \) of the domain \( \Omega \) we consider the following linearized error problem [6]:

\[
\begin{align*}
\text{given } & \mathcal{T}_h = \{ K \}, u_h \in V_h; \\
\text{find } & e := e_h \in E, V = V_h \oplus E \text{ such that } \\
& b_\Omega(u_h; e, v) = \rho_h(u_h; v) \quad \forall v \in E, \text{ where } \\
b_\Omega(w; z, v) &= a_\Omega(z, v) - \int_\Omega f[u]wz \, dx \\
& \rho_h(w; v) := n_\Omega(w; v) - a_\Omega(w, v) \quad \forall w, z, v \in V.
\end{align*}
\]

(3)

To solve the problem (3) approximately, we apply the Galerkin method and in the finite dimensional subspace \( E_h \subset E \) we find the approximation \( e_h(e_h(x)) \) such that \( e_h(x) = \sum_{K \in \mathcal{T}_h} e_h(x) = \sum_{K \in \mathcal{T}_h} \lambda_K \phi_K(x) \in \mathcal{V} \). The functions \( \phi_K = \phi_K(x) \) generate an orthogonal basis [6] of the subspace \( E_h \subset E \) and satisfy the properties: \( \text{supp } \phi_K = K \quad \forall K \in \mathcal{K}, \phi_K(A_i) = 0 \quad \forall A_i \in K, \text{ where } A_i \text{ are vertices of } K \).

Consequently, for each finite element \( K \in \mathcal{T}_h \) we can formulate the local error problem and calculate

\[
\begin{align*}
e_h(x) &= \lambda_K \phi_K(x) = \frac{\rho_K(u_h; \phi_K)}{\rho_K(u_h; \phi_K)}(x)
\end{align*}
\]

(4)

We select the following basis functions for triangular finite elements with the barycentric coordinates \( \{ L_m(x) \}_{m=1}^{\infty} \):

\[
\begin{align*}
\phi_K^{PL} &= 27L_1L_2L_3, \\
\phi_K^{NV} &= 3[L_1L_2 + L_2L_3 + L_3L_1], \quad \forall K \in \mathcal{T}_h,
\end{align*}
\]

(5)

to construct Dirichlet \( e_h^{PL} \) and Neumann \( e_h^{NV} \) AEEs of linear finite element approximations correspondingly. For other details of the \( h \)-adaptive FEM construction see [5].
4. Numerical results

In the numerical analysis we solve the singularly perturbed and semi-linear boundary value problems with explicit exact solutions using the uniform and local adaptive mesh refinement. We use the Dirichlet estimator in an adaptive criterion and the bisection method for the local mesh refinement, see [6].

The singularly perturbed problem with internal layer is stated as follows

\[-10^{-3} \Delta u + (\beta_1, \beta_2) \nabla u = 0 \quad \text{in} \quad \Omega = [0, 1]^2, \]

where $\beta_1(x) = x_1 - 0.6, \beta_2(x_2) = x_2 - 0.3$. The solution $u_{ext}(x, y) = G[m\beta_1(x_1) + \psi_2(x_2)]G[m\beta_2(x_2) - \psi_2(x_1)], m = \cos(\pi/6), v = \sin(\pi/6), G(z) = \frac{1}{z} + \frac{1}{z} f(z/\sqrt{2})$.

We solve problem (6) using linear FEM approximation on the uniform triangular meshes.

Finally, we consider the following semi-linear problem

\[-\Delta u = a(r)u^3 + b(r)u^2 \quad \text{in} \quad \Omega = [0, 1]^2,\]

\[u = u_{ext} \quad \text{on} \quad \Gamma_D, \nu \nabla u = 0 \quad \text{on} \quad \Gamma_N,\]

where $\Gamma_D = \{x \in \Omega : x_1 = 1\} \cup \{x \in \Omega : x_2 = 1\}, \Gamma_N = \{x \in \Omega : x_1 = 0\} \cup \{x \in \Omega : x_2 = 0\}, a(r) = -8f^2 \cos^2 r^2, b(r) = 4f^2 \cos^2 r^2 \sin r^2$, the solution $u_{ext} = [\sin r^2 + 2]^{-1}$.

Numerical results for the problem (7) are presented in tables 3, 4. Note that tables 2, 4 show only selected rows.

Finally, the suggested Dirichlet and Neumann AEEs can be extended to the bilinear FEM approximations, for details see [7], by choosing the following basis functions instead of (5)

\[\phi_{Dir}(\alpha, \beta) = (1 - \alpha^2)(1 - \beta^2), \quad \phi_{Neu}(\alpha, \beta) = 1 - \frac{1}{2}(\alpha^2 + \beta^2), \quad \forall Q \in \mathbb{H}_h.\]

The local coordinates of a quadrilateral $Q \in \mathbb{H}_h$.

References


A fast multipole Boundary Element Method in the analysis of 3D linear elastic structures

Jacek Ptaszny
Institute of Computational Mechanics and Engineering, Silesian University of Technology
Konarskiego 18A, 44-100 Gliwice, Poland
e-mail: jacek.ptaszny@polsl.pl

Abstract

A fast multipole boundary element method for 3D elasticity problem was developed by the application of the fast multipole algorithm and isoparametric 8-node boundary elements with quadratic shape functions. The problem is described by the boundary integral equation involving the Kelvin solutions. In order to keep the numerical integration error on appropriate level, an adaptive method with subdivision of boundary elements into subelements, described in the literature, was applied. Efficiency of the method is illustrated by numerical examples involving structures with high stress concentration and relatively high number of degrees of freedom.

Keywords: linear elasticity, fast multipole boundary element method, quadratic elements, adaptive integration

1. Introduction

In the paper, application of the boundary element method (BEM) in the analysis of 3D linear elastic structures is considered. The most important advantage of the BEM in relation to domain methods is discretization only of boundary [1, 2, 8, 9]. This feature makes the boundary element method efficient in the solution of many problems, e.g. in micromechanics [5, 6, 10, 11, 12, 13, 14]. One of the modern versions of the method, that was applied in this work, is the fast multipole BEM (FMBEM) [9]. It is characterized by a reduced computational complexity in comparison with the conventional BEM.

2. Fast multipole boundary element method

The idea of FMBEM is based on hierarchical grouping and distribution of potentials. The potentials are boundary integrals occurring in the integral equation of the considered problem [1, 2, 8, 9]. This idea is realized by the application of multipole expansions for far-field potentials (Fig. 1). Coefficients of the expansions – multipole moments – are dependent on quantities related to clusters of integration points. The centers of the multipole moments are shifted to centers of larger groups (moment-to-moment translation, M2M) and transformed into moments of local expansion (multipole-to-local translation, M2L). Centers of the local moments are shifted to centers of smaller clusters (local-to-local translation, L2L) and to each collocation point. The near-field potentials are calculated directly as in the conventional BEM. The boundary elements are hierarchically clustered within regions that forming a tree structure. A detailed description of the FMBEM can be found in the book [9].

An FMBEM code for the analysis of 3D problems of linear elasticity was developed. The method is based on the direct boundary integral formulation with Kelvin’s fundamental solution. To the boundary discretization, 8-node isoparametric Serendipity elements, with adaptive subdivision, are applied [1, 4]. The subdivision is based on the determination of the number of Gauss quadrature points for one integration direction. The number was determined experimentally in the paper [4] in order to keep the relative integration error less than $10^{-3}$. The number depends on the characteristic length of the element of integration and the distance from the collocation point to the integration point. If the number of Gauss points exceeds four, element is subdivided in corresponding direction.

The multipole and local expansions of integral kernels involve spherical harmonics [9]. The system of equations is solved by using preconditioned GMRES.

3. Numerical examples

3.1. Infinite solid body with two cavities

An infinite solid body with two spherical cavities was analysed. The solid was loaded uniaxially in the vertical direction. The structure was modelled by superposition of two problems: an infinite homogeneous solid body under the uniaxial stress state, and a solid body with two cavities loaded on their boundaries by tractions induced by the uniaxial stress with opposite sign [1, 6, 15]. Using the BEM, the structure can be modelled as an infinite domain problem without any artificial boundary (the model is loaded on the boundaries of the
cavities). Fig. 2 shows the normal stress distribution in the vertical direction. The numerical solution was compared to the analytical results given in [7]. By using the developed FMBEM, accuracy of 1% or 0.01% can be achieved, dependently on the position of a point at the equator of the cavity, when the number of degrees of freedom (DOF) is \( O(10^4) \).

![Figure 2: The distribution of normal stress component in the vertical direction](image)

3.2. Structures with many spherical cavities

A representative volume element (RVE) of a material with cubic arrangement of cavities was modelled. The RVE was a cube with 5×5×5 array of spherical cavities. Porosity of the model was equal to 0.4. Fig. 3 shows the interior of the RVE with boundary elements. The number of DOF exceeds 120 000. On the external boundary (the cube) the displacement boundary conditions were imposed [16]. The averaged stresses were computed using the resulting traction forces on the cube faces. At the macroscale, the material is characterized by cubic symmetry, which is the simplest case of anisotropy. As a result of homogenization, the overall bulk modulus and two shear moduli were obtained. The results were compared to the corresponding analytical models [3]. The numerical results are in agreement with the models. The computations were performed by means of an unremarkable laptop PC (Intel Core i3 CPU, 2.40 GHz, 8 GB RAM). The analysis time was about 10 hours.

![Figure 3: Discretized model of the RVE](image)

4. Conclusions

An FMBEM for the analysis of 3D linear elastic structures was developed, coupling the direct formulation with a higher order approximation scheme and the fast multipole method. In order to minimize the numerical integration error, the procedure of adaptive integration with division into subelements, available in the literature was adopted. Two numerical examples were presented. All results agreed with the corresponding analytical models. Future investigations may involve comparison between the FMBEM and the finite element method (FEM) in terms of accuracy and efficiency.

References

An adaptive Finite Element Method for contact problems in finite elasticity

Waldemar Rachowicz1,*, Adam Zdunek2, Witold Cecot1*

1Institute of Computer Science, Cracow University of Technology
ul. Warszawska 24, 30-155 Kraków, Poland
e-mail: wrachowicz@pk.edu.pl
2Swedish Defense Research Agency FOI
SE-164 90 Stockholm, Sweden
e-mail: zka@foi.se
3Institute for Computational Civil Engineering
ul. Warszawska 24, 30-155 Kraków, Poland
e-mail: plcecot@csf-kr.edu.pl

Abstract

We consider a mortar finite element method for 3D finite deformation frictionless contact problems proposed by Popp et al. [4] in a version enabling error estimation and mesh adaptivity. Contact of elastic bodies made of hyperelastic nearly incompressible materials is considered. Mixed formulation of finite elasticity with contact is approximated with hexahedral elements. Consistent linearization allows one to apply the Newton-Raphson method to solve the nonlinear problem. Error estimation and adaptivity of finite element meshes allow to reduce the error of approximation.

Keywords: finite element method, mortar, contact, Lagrange multipliers, finite deformations, frictionless

1. Introduction

Contact problems constitute a large part of engineering applications. Efficient algorithms for modeling contact are still under investigation of many researchers. In this paper we focus our attention on the so-called mortar method with Lagrange multipliers as a technique to enforce the no-penetration condition of bodies in contact as proposed by Popp et al. [4]. The mortar algorithm, also referred to as a segment-to-segment approach (STS), enforces the no-penetration condition in a weak form as opposed to the widely spread node-to-segment method (NTS) [3, 6] enforcing this condition in a pointwise fashion (collocation). The mortar method has many advantages when compared to NTS: it satisfies the contact patch test, shows better robustness in the case of finite deformations and finite sliding problems.

Advantages of mortar method are also profoundly manifested in the context of adaptive methods being used for approximation. Namely, collocation procedure of NTS frequently fails if densities of elements on both contacting surfaces differ substantially as then the algorithm reflects the well-known fact that the pointwise Dirichlet boundary condition is ill-posed in 3D elasticity. In contrary, the mortar method uses smoothly distributed Lagrange multipliers to model the contact forces. Needless to say that this enables one to apply the standard error estimation techniques and gives almost unlimited freedom in generating density of meshes on contact surfaces. Finally, a version of segment-to-segment technique with order of approximation \( p = 2 \) has been investigated which is not the case for the node-to-segment approach.

Having said all this in favour of the STS technique we should mention the price to be paid for the quality of the method: it is the necessity of performing a fairly complex integration procedure over intersections of finite elements of two surfaces being in contact. It also involves relatively complicated linearization procedures of the contact problem.

2. Mortar approach

We adopt a popular terminology in contact mechanics calling contact surface \( \Gamma^{(1)} \) the slave and \( \Gamma^{(2)} \) the master surface. Assuming material description of deformation we denote current locations \( x_i \) of points on \( \Gamma^{(i)} \) as functions of the reference locations \( X_i \) and time \( t \): \( x_i = x_i(X_i, t) \). We define the gap function \( g(X, t) \) indicating the possibility of penetration of contacting surfaces as follows. Let \( \hat{x}^{(2)} \) be the projection of slave point \( x^{(2)} \) onto the master surface along the unit normal vector \( n(x^{(1)}(X^{(1)}, t)) \). Then

\[
g(X^{(1)}, t) = -n \cdot (x^{(1)} - \hat{x}^{(2)}).
\] (1)

We express the negative contact tractions \( z \) on the slave surface \( \Gamma^{(1)} \) by their cartesian coordinates in the natural orthogonal basis of vectors \( (n, \tau^\ell, \tau^\gamma) \):

\[
z = z_n n + z_\tau^\ell \tau^\ell + z_\tau^\gamma \tau^\gamma
\] (2)

With this notation the frictionless contact can be formulated as follows:

\[
\begin{aligned}
\delta &\geq 0, &\delta(z_n) := \int_{\Gamma^{(1)}} g\delta z_n dS, \forall \delta z_n \geq 0, \\
z_n &\geq 0, &z_\tau^\ell = 0, &z_\tau^\gamma = 0
\end{aligned}
\] (3)

where the no-penetration condition (3)_1 was written here in the strong and weak versions, respectively. After introducing the FEM discretization the collocation conditions (3)_2,\ldots,5 for the node \( x_j \) and the variational form of (3)_1 corresponding to the shape

* W. Rachowicz and W. Cecot acknowledge the financial support via grant No. UMO-2011/01/B/ST6/07306 received from the Polish National Center of Science.
function of node $x_j$ can be written as follows:
\[
\begin{align*}
\tilde{g}_j & \geq 0, \\
(z_n)_j & \geq 0, \\
(z_n)_{	ilde{g}_j} & = 0, \\
(z_\tilde{g}_{j}) & = 0, \\
(z^2_{j}) & = 0.
\end{align*}
\]
In addition, we may formulate the involving inequality conditions (4)_{1-3} by an equality condition expressed by a specially designed function $C_j := z_{n,j} - \max(0, z_{n,j} - c_n g_j), c_n > 0$:
\[
\begin{align*}
\tilde{g}_j & \geq 0, \\
(z_n)_{j} & \geq 0, \iff C_j(z_{n,j}, \tilde{g}_j) = 0
\end{align*}
\]
which allows to express linearization of contact problem in a classical way. This combined with the standard linearization of finite elasticity constitutes the final formulation of the contact problem.

3. FE approximation, error estimation and adaptivity

In our applications we use a nearly incompressible hyperelastic material possibly reinforced with fibres. In such situation a mixed formulation of the finite elasticity problem is usually applied, with displacements and pressure $(u, p)$ as unknowns which involves the principle of virtual displacements (expressing equilibrium) and the weak enforcement of the constitutive relation for the pressure. We use a family of hexahedral elements with trilinear approximation for displacements and constant pressures (though higher order approximation is available). Quadratic approximation of contact displacements is under development. Continuity of approximation between the elements of uneven size (due to adaptivity) is enforced via constrained approximation. According to results of Rüter and Stein [5] one can express a posteriori error estimate of the FEM approximation of the mixed formulation by the residual error estimate corresponding to the principle of virtual displacements. We evaluate it with two kinds of residual techniques: element residual method of Bank and Weiser [2] and a method of self-equilibrated residuals proposed by Ainsworth and Oden [1]. The first of the techniques is faster, the second one is more reliable. In implementations of the residual error estimates the Lagrange multipliers on surfaces being in contact are considered as the known surface tractions of the residual error estimates the Lagrange multipliers. Integration of the corresponding work of virtual displacements is performed according to mortar procedures. The strategy of adaptivity of finite element meshes is based on the principle of equidistribution of errors. In practice this boils down to subdividing the elements with the largest estimated errors.

4. Examples

We test the mortar contact algorithm on two example problems. In the first of them a half-cylindrical die of the hyperelastic almost incompressible material with the shear modulus $\mu_1 = 1.0$, and with the internal and external radii $r_1 = 2.8$, $r_2 = 3.0$ is intruding into an elastic block of dimensions $4 \times 6 \times 3.0$ with the shear modulus $\mu = 0.002$. The vertical displacement of the die is $1.4$. Figure 1 presents the map of vertical displacements on the deformed configuration.

The second test concerns pressurization of an elastic hollow sphere of internal and external radii $r_1 = 4.0$, $r_2 = 4.5$ and the shear modulus $\mu = 0.002$, placed inside a cylindrical tube of internal radius $r_2$ and external radius $r_3 = 4.67$. The length of the tube is $l = 8.0$, the shear modulus $\mu = 0.0001$, and the sphere is located in the middle of the cylinder. The value of the pressure is $p = 0.00018$. Due to the symmetry we solve only $1/8$ of the structure using appropriate symmetry boundary conditions. Figure 2 presents the map of radial displacements on the deformed configuration.

References


Numerical modeling of thermopiezoelectricity steady state forced vibrations problem using adaptive Finite Element Method

Vitaliy Stelmashchuk¹, Heorhiy Shynkarenko²

¹Faculty of Applied Mathematics and Informatics, Ivan Franko National University of Lviv
Universytetska 1, 79000 Lviv, Ukraine
e-mail: vistelm@gmail.com

²Faculty of Production Engineering and Logistics, Opole University of Technology
76 Prószkowska Street, 45-758 Opole, Poland
e-mail: h.shynkarenko@po.opole.pl

Abstract

The purpose of the research is to build a h-adaptive FEM scheme for solving a kind of thermopiezoelectricity problems. We use a linear mathematical model of thermopiezoelectricity and construct the initial boundary value problem. Then a case of steady-state forced vibrations of pyroelectricity is considered. After formulation of the corresponding variational problem, variable separation and applying Galerkin classic procedure, a numerical scheme is obtained for solving the problem. Finally, a posteriori error estimators and h-adaptive scheme for thermopiezoelectric forced vibration problem are built. Numerical experiments are set for PZT-4 pyroelectric bar under influence of different types of loadings.

Keywords: thermopiezoelectricity, forced vibrations, Galerkin discretization, a posteriori error estimator, h-adaptive FEM, pyroelectric bar, PZT-4 ceramics

1. Introduction

Many modern engineering applications and devices utilize the piezoelectric and pyroelectric materials, see Ref. [6, 9, 10]. In our research we consider only the case of thermopiezoelectric steady state forced vibrations. Also our aim is to achieve solution accuracy on some preset level. Numerical scheme based on h-adaptive FEM will be built to do so.

2. Thermopiezoelectricity steady state forced vibrations problem

General thermopiezoelectricity problems were firstly described in Ref. [4, 5]. A comprehensive review of the studies on this topic can be seen in Ref. [1]. Let us suppose that a pyroelectric specimen occupies the domain Ω in Euclidean space R^n, where n = 1, 2 or 3. In accordance with Ref. [3, 7, 8], we assume that thermopiezoelectric behaviour of specimen can be quite fully described by an elastic displacement vector u = \{u_i(x, t)\}_{i=1}^n, electric potential p = p(x, t) and temperature θ = θ(x, t), which satisfy the system of coupled partial differential equations of motion, electrodynamics and heat conduction. The specific case of the above problem is a steady state of forced vibrations with angular frequency ω = const > 0. In this case, (u(x, t), p(x, t), θ(x, t)) = (U(x), P(x), Θ(x))e^{-iωt}, where U(x), P(x) and Θ(x) are the unknown amplitudes of displacement, electric potential and temperature, respectively. After introducing the space of admissible amplitudes W, the variational form of the thermopiezoelectricity problem can be represented as in Eqn (1):

\begin{align}
\int_{\Omega} \left\{ \begin{array}{l}
given\ angular\ frequency\ \omega = const > 0, \chi \in W; 
find\ vector\ \psi = (U(x), P(x), Θ(x)) \in W\ such\ that\ \varphi \in W, 
\end{array} \right. \\
\varphi \in W.
\end{align}

The bilinear form Ω(\cdot, \cdot) : W × W → R is the linear combination of several bilinear forms, which are built on the basis of physical properties of pyroelectric (mass, stiffness, piezoelectricity, pyroelectricity coefficients, etc.). The linear functional χ : W → R is the linear combination of mechanical, electrical and heat loadings. The detailed expressions and physical meanings can be found in Ref. [7].

3. Galerkin discretization and obtained numerical scheme

Let Ω be a partition of domain Ω into non-overlapping shape regular elements, and W is a space of finite element approximations based on that partition. Then after applying standard Galerkin procedure, we get the following system of linear algebraic equations for calculation of nodal values of finite element approximation for the unknown amplitudes:

\begin{align}
\begin{bmatrix}
\omega A & -D & 0 & E^T & 0 & Y^T \\
D & \omega A & -E^T & 0 & -Y^T & 0 \\
0 & \omega E & Z & \omega G & 0 & \omega B^T \\
-ωE & 0 & -ωG & Z & -ωB^T & 0 \\
0 & \omega Y & 0 & ωB & K & ωS \\
-ωY & 0 & -ωB & 0 & -ωS & K
\end{bmatrix} \times
\begin{bmatrix}
U_1 \\
U_2 \\
P_1 \\
P_2 \\
Θ_1 \\
P_3
\end{bmatrix} =
\begin{bmatrix}
L_1 \\
L_2 \\
R_1 \\
R_2 \\
F_1 \\
F_2
\end{bmatrix}^T.
\end{align}

Here the unknown vectors U_1, U_2, P_1, P_2, Θ_1, Θ_2 are the nodal values of amplitudes (real and imaginary parts) of elastic displacement, electric potential and temperature respectively. Matrices M, C, A, E, Y, Z, G, B, K, S are mass, stiffness, viscosity, piezoelectricity, thermal expansion, electric conductivity, dielectric susceptibility, pyroelectricity, thermal conduction and heat capacity matrices respectively, while matrix D = ω^2M + C. Vec-
tors \( L_1, L_2, R_1, R_2, F_1, F_2 \) represent amplitudes of mechanical, electrical and heat loadings.

4. A posteriori error estimator

Similarly to Ref. [2], with respect to variational problem (1) and its discretized form, we can easily formulate the variational problem for finding an approximation error:

\[
\left\{ \begin{array}{l}
given \omega = \text{const} > 0, \psi_h \in W_h; \\
\text{find error } e = \psi - \psi_h \in E = W \setminus W_h \text{ such that} \\
\Pi_\omega (e, w) = \langle \rho_\omega(\psi_h), w \rangle := \\
\quad : = \langle \chi, w \rangle - \Pi_\omega (\psi_h, w) \quad \forall w \in E.
\end{array} \right.
\] (3)

To solve the problem (3), we apply a Galerkin procedure using a certain finite dimensional subspace \( E_h \subset E \):

\[
\left\{ \begin{array}{l}
given \omega = \text{const} > 0, \psi_h \in W_h; \\
\text{find error estimator } e_h \in E_h \text{ such that} \\
\Pi_\omega (e_h, w) = \langle \rho_\omega(\psi_h), w \rangle \quad \forall w \in E_h.
\end{array} \right.
\] (4)

5. Adaptive FEM

Each iteration of adaptive scheme brings about an error indicator \( \eta_K \) for each finite element \( K \) of partition \( \mathcal{K} \) by the following rule:

\[
\eta_K = \sqrt{N \| e_h \|_{1,K} ^2} \quad \forall K \in \mathcal{K},
\] (5)

where \( N = \text{card} \mathcal{K} \) is the amount of finite elements in conforming partition \( \mathcal{K} \). If \( \eta_K > 1 \) occurs on any finite element \( K \) (the error is greater than average on partition), the element \( K \) is then refined. If a total error indicator \( \eta = \| e_h \|_{1,\mathcal{K}} \leq 100\% \) is less or equal to a given admissible error level \( \delta \), the algorithm stops.

6. Numerical experiment

Various singularities often occur while modeling heat loadings applied to the pyroelectric specimen. Therefore, we consider the following numerical experiment. A PZT-4 (physical properties can be seen in Ref. [8, 10]) ceramic bar with the length \( L = 0.01 \text{ m} \), loaded on the right edge with a heat flux \( \tilde{q}(x, t) = 100 \cdot \cos \omega t \cdot \text{J.m}^{-2} \cdot \text{s}^{-1} \), where angular frequency \( \omega = 3 \cdot 10^8 \text{ rad.s}^{-1} \). The left edge is fixed, grounded and with constant temperature. We start with the uniform mesh of \( N = 256 \) finite elements with a piecewise-linear approximation of solution. The solutions contain oscillations, see Fig. 1, and the relative error is equal to 32.48%.

An adaptive scheme with an posteriori error estimator, which uses quadratic bubble-function, is then applied. The preset level of accuracy is \( \delta = 0.01\% \). It took 12 iterations of the adaptation process to achieve the goal. The final mesh consists of 410 finite elements. Figure 2 presents the relative error convergence.

7. Conclusions

The constructed \( h \)-adaptive scheme allows to solve the thermopiezoelectricity forced vibrations problem with a preset level of accuracy. Since velocity of heat transfer and mechanical impulse propagation speed are of different scales, singularities are often noticed, when heat loadings are applied, and it is impossible to obtain sufficient solutions without using a \( h \)-adaptive scheme. The numerical experiment described shows the scheme in action.

References

Application of the element residual methods to dielectric and piezoelectric problems

Grzegorz Zboiński*
Institute of Fluid Flow Machinery, Polish Academy of Sciences
Fiszera 14, 80-231 Gdańsk, Poland
e-mail: zboi@imp.gda.pl
Faculty of Technical Sciences, University of Warmia and Mazury
Oczapowskiego 11, 10-719 Olsztyn, Poland

Abstract

The paper deals with some theoretical and implementation aspects of application of the element residual methods (ERM) to dielectric and piezoelectric problems. In particular the residual equilibration (or equilibrated residual) method (REM) is a subject of our interest. The aspects presented in the paper are still under investigation. In the conducted research we focus on the upper bound properties of REM in the dielectric and piezoelectric cases. Furthermore the algorithms of the method for both cases are of our interest, especially the details concerning the equilibration process and the solution of REM element local problems. The convergence properties for both types of problems, and for the mechanical elasticity problem as well, are the base for determination of the algorithmic details in the piezoelectric case. Also, the robustness and effectiveness of the method are subjects of our research in both cases. These two performance aspects are compared to the elasticity case.

Keywords: dielectricity, piezoelectricity, error estimation, element residual methods

1. Introduction

This research is motivated by the present interest in application of the piezoelectric members as actuators and sensors in the so-called intelligent structures and structural health monitoring, respectively. Additionally, the questions of application of the adaptive finite element methods to dielectric and piezoelectric problems is in its early stage of theorisation and implementation. These questions constitute the second motivation of this research.

In the paper we focus our attention on error estimation in the adaptive modelling and adaptive finite element analysis of the dielectric and piezoelectric problems. The main question we would like to answer in this research is if the element residual methods in general, and the residual equilibration method in particular, can be applied to two mentioned types of problems. Additionally, we detect how the theory of the error estimation of the elliptic problems, and the specific elasticity problem as well, has to be modified in the case of the dielectricity and piezoelectricity. We are also interested in the modifications of the existing residual equilibration algorithms in the case of the purely electric (dielectricity) and coupled electro-mechanical (piezoelectricity) cases. Such algorithms were applied successfully to the adaptive modelling and analysis of complex elastic structures [1].

2. State of the art

We address two issues in this brief survey. The first one is adaptive modelling and analysis of dielectric and piezoelectric media or systems. The second one concerns application of the element residual methods to error estimation in the electric and coupled electro-mechanical problems.

2.1. Adaptivity in dielectric and piezoelectric systems

It should be boldly stated that the adaptive analysis of dielectric and piezoelectric systems is not very popular and only a few papers address the issue. Many of the available papers on the topic have preliminary character and are published in conference proceedings. Among the papers on numerical analysis of the piezoelectric media we mention the works [2] and [3], which present the classical (non-adaptive) calculations via h-method and parametric studies with p-method, respectively. The rare example of the application of the adaptive approach to the analysis of piezoelectric problems can be found in [4], where the h- and p-methods are applied. The suggestions concerning application of the hp-adaptivity to the dielectric and piezoelectric problems can be found in our conference and post-conference papers, i.e. [5, 6], where the hp-adaptive finite element method as well as the hierarchical dielectric and piezoelectric models are proposed, respectively. The related comparative h- and p-convergence studies for the model dielectric and reference elastic problems are presented in [7], where the resultant conclusions on how the hp-mesh adaptation should be performed in the case of piezoelectrics.

2.2. Error estimation for dielectricity and piezoelectricity

No examples of the direct application of the element residual methods (ERM) and the residual equilibration method (REM) to the dielectric and piezoelectric problems can be found in the widely available literature. The suggestion on the application of REM to dielectrics is presented in the conference abstract [8] only. Some considerations concerning the application of REM to piezoelectric systems are present in [5].

Even though the mentioned methods are not common in the case of dielectricity and piezoelectricity, their application to elastic and other elliptic problems is a standard. The REM in the general context of elliptic problems was presented by Ainsworth and Oden [9, 10]. The 3D-elasticity case is described in [11]. The case of conventional shell-like and plate-like elastic structures was described in [12], while the application of the residual equilibration method to 3D-based complex elastic structures was proposed in [1]. The mentioned works can be a direct hint and justification for the application of the REM to dielectric problems due to its similarity to elliptic problems in general, and the

*The partial support of the Polish Scientific Research Committee (KBN) and the Polish National Science Centre (NCN) under the grant no. 5153/T02/B/2011/40 is thankfully acknowledged.
elastic problem in particular. The case of piezoelectricity is much more difficult and needs special and careful treatment due to its electro-mechanical, coupled character.

3. Application of REM to dielectricity and elasticity

It should be noticed that we deal with formal similarity of formulations of the problems of elasticity and dielectricity. This similarity is reflected by the analogy in the local, variational and finite element formulations of both problems. Hence, the available theoretical and implementation findings concerning elasticity can be easily extended onto the dielectricity case.

3.1. Bounding features of the estimation with REM

It is well known that the residual equilibration method, constituting the most effective example of the element residual methods, possesses the upper bound property of the global approximation error estimate, when applied to elliptic problems of the second order (see [9, 10]). This general case includes also the linear elasticity problems of various type (compare [11, 12] and [1]). These results can be extended onto dielectric problems as well.

3.2. The equilibration process of REM

Equilibration of the residuals performed in the case of elasticity can be adopted to the dielectric case. Such adoption needs replacement of the equilibrated interelement stress vectors with the equilibrated scalar electric charges. It also requires replacement of the mechanical quantities (stresses, strains, displacements and forces) with the electric ones (electric displacements, electric fields, electric potential and electric charge, respectively).

3.3. REM local problems

The REM local problems of dielectricity can be defined in analogy to the elasticity case, i.e. the local element equations describing element mechanical equilibrium have to be replaced with the corresponding equations describing electric equilibrium of the element. In particular, the element stiffness and load vectors, including the load from the equilibrated interelement stresses, have to be changed for the dielectric matrix and the charge vector, composed of the external and internal (equilibrated) charges.

4. Application of REM to piezoelectricity

Here we consider generalisation or extension of the residual equilibration method onto piezoelectric media.

4.1. The bounding property of the estimation

The proof of the bounding property of the global approximation error estimate for the general piezoelectric problem, in a sense applied to elasticity and dielectricity, is not possible due to a coupling of the potential energies of the electric and mechanical fields. We deal with subtraction of both energies, i.e. the mechanical one is converted into the electric one or vice versa.

4.2. The equilibration process

There are no formal obstacles to obtain the equilibrated interelement stresses and the interelement electric charge for the electro-mechanically coupled piezoelectric problems. This needs inclusion of the coupling terms (resulting from the coupling part of the potential energy) into the equilibration equations corresponding to the elasticity and dielectric cases.

4.3. The local problems of REM

The coupled local problems can be formulated in the similar way, i.e. by addition of the coupling terms to the local equations corresponding to the elasticity and dielectricity cases. Some simplifying decoupled options also exist. However, their applicability needs theoretical substantiation and numerical verification.

5. Conclusions

Algorithmic extension of the residual equilibration method onto the dielectric and piezoelectric cases is possible. However, the upper bound property of the estimation can be proved for the dielectricity case only.

References


MS02

Axially Moving Structures

organized by Y. Vetyukov and M. Krommer
Elasto-plastic bending of steel strip in a hot-dip galvanizing line

Michael Baumgart1, Andreas Steinboeck2, Martin Saxinger3, and Andreas Kugi4
1,3,4Christian Doppler Laboratory for Model-Based Control in the Steel Industry, Automation and Control Institute, Vienna University of Technology, Gußhausstraße 27-29, 1040 Vienna, Austria
e-mail: baumgart@acin.tuwien.ac.at1, saxinger@acin.tuwien.ac.at3, kugi@acin.tuwien.ac.at4
2Automation and Control Institute, Vienna University of Technology, Gußhausstraße 27-29, 1040 Vienna, Austria
e-mail: andreas.steinboeck@tuwien.ac.at

Abstract

A quasi-static, first-principle model of axially moving steel strip in a continuous hot-dip galvanizing line is presented. The model yields the bending line of the strip and takes the history of plastic deformation into account. Numerical integration of the material model of elasto-plastic deformation is algorithmically separated from the solution of the boundary value problem of the bending line. Thus, the influence of different roll settings on the displacement of the strip at upstream tools like gas wiping dies can be efficiently analyzed.

Keywords: strip processing lines, hot dip galvanization, plastic deformation, deformation history, axially moving strip

1. Introduction

In continuous hot-dip galvanizing lines, cf. Fig. 1, the strip – subject to tension and high temperatures – may be bent plastically at the deflection rollers of the plant. These bends cause deviations of the strip from the ideal flat shape, i.e., a coil-set or a crossbow.

![Figure 1: A hot-dip galvanizing line](image)

Just above the zinc bath, the strip passes the so-called gas wiping dies, which remove excess zinc and control the thickness and uniformity of the zinc coating. The crossbow can lead to an inhomogeneous coating, so-correction and stabilization rolls should effectively reduce this shape defect by means of controlled plastic bending of the strip. Moreover, these rolls should compensate the position deviation caused by the coil-set.

In order to optimally control the position of the correction roll, the stabilization roll and the gas wiping dies, a computationally efficient mathematical model of the shape in terms of the transversal deflection of the strip is required. A principal challenge of modeling the strip in the hot-dip galvanizing line is that the history of elasto-plastic deformation of each material point has to be conceptually moved with the strip through the plant. Furthermore, material models for plastic deformation are generally nonlinear and have to be numerically integrated for given deformation increments. Moreover, at the correction and the stabilization roll, the degree of deformation in terms of the strip curvature is a priori unknown. Generally, the problem can be solved by means of fully discretized FE models, which usually entail high computational costs.

In the paper, a quasi-static, first-principle model of the elasto-plastic bending line of the strip is presented. In order to predict the mean strip displacement at the gas wiping dies, the model focuses on the one-dimensional transversal displacement along the direction of strip motion \(x\). The history of plastic deformation up to the stabilization roll is systematically taken into account. Moreover, numerical integration of the model of elasto-plastic deformation is algorithmically separated from the calculation of the actual bending line. This way the influence of different roll settings on the strip displacement can be efficiently analyzed.

2. Quasi-static model of a strip under tension

Figure 2 shows configuration of the rolls and the considered domain of the strip, which touches the rolls at the a priori unknown locations \(x_\rho\), \(\rho \in \{BR, CR, SR, TR\}\). The strip is subjected to a constant tensional force \(N\). Inertia effects and the influence of the strip velocity on the quasi-static solution are neglected.

The balance of forces and moments at an infinitesimal strip element yields the differential equation for the quasi-static displacement \(w(x)\) in \(z\)-direction [2],

\[ M''(x) + Nw''(x) + q(x) = 0, \]

where the spatial partial derivative is denoted by \((\cdot)' = \partial(\cdot)/\partial x\). Due to the elasto-plastic deformation, the bending moment \(M(x)\) is a nonlinear function of the strip curvature \(\kappa(x) \approx w''(x)\), cf. Sec. 3. With \(q(x)\), transversal loads induced by electro-magnetic actuators or cooler arrays can be considered.

---

1 Financial support by the Austrian Federal Ministry of Science, Research and Economy, the National Foundation for Research, Technology and Development, and voestalpine Stahl GmbH is gratefully acknowledged.
2 The second author gratefully acknowledges financial support provided by the Austrian Academy of Sciences in the form of an APART-fellowship at the Automation and Control Institute of Vienna University of Technology.
The boundary value problem (BVP) Eq. (1) is complemented by the following boundary and interface conditions: The deflection is prescribed by the roll surface at all contact points. At \( x \) the curvature and the bending moment are given by the roll curvature and the previous deformation history, respectively. At the intermediate rolls, \( i.e., x \in \{ x_{CR}, x_{BR} \} \), the slope, the curvature, and the bending moment have to be continuous. The actual boundary conditions and the contact point at the tower roll are expected to have only minor influence on the strip displacement near the zinc bath rolls because the tower roll at \( x_{TR} \) is far away from the stabilization roll. Here, \( w'(x_{TR}) = 0 \) is set.

3. Material model of elasto-plastic strip bending

A common model describing the incremental elasto-plastic deformation are the Prandtl-Reuss equations, cf. [1]. Here, the von Mises yield criterion with the yield stress \( \sigma_{yld} \) is adopted. In the zinc bath and the upstream process steps, the strip is exposed to temperatures greater than \( 400^\circ\text{C} \) and thus, it starts to yield at reduced yield stresses compared to room temperature. For plane stress conditions and vanishing shear deformation, nonlinear relations between the longitudinal strain increment \( d\varepsilon_{xx} \) and the stress increments \( d\sigma_{xx} \) and \( d\sigma_{yy} \) at a material point can be found for the plastic domain in the general form

\[
d\sigma_{xx} = g_{p,x}(\sigma_{xx}, \sigma_{yy}, \sigma_{yld})d\varepsilon_{xx} \tag{2a}
\]

\[
d\sigma_{yy} = g_{p,y}(\sigma_{xx}, \sigma_{yy}, \sigma_{yld})d\varepsilon_{xx} \tag{2b}
\]

These relations may also include effects like work hardening. For the elastic domain, Hooke’s law yields relations \( g_{e,x} \) and \( g_{e,y} \) that are independent of the total stresses.

If the mean tensile stress due to the tensile force \( N \) is significantly smaller than \( \sigma_{yld} \), it can be neglected. The strain increments can then be expressed in terms of the strip curvature \( \kappa \), \( i.e., d\varepsilon_{xx} = -\kappa dx \) in Eq. (2). All strains and stresses are assumed to be constant with respect to \( y \). Starting from a known initial value \( \kappa_0 \) and a stress profile \( \sigma_{xx,0} \) along the strip cross section of width \( b \) and height \( h \), the stress resultant \( M \) can be obtained by integration, \( i.e.,

\[
M = b \int_{-b/2}^{b/2} \left( \int_{\kappa_0}^{\kappa} -\kappa_0 - \kappa dx \right) d\varepsilon_{xx} + b \int_{-b/2}^{b/2} \sigma_{xx,0} d\varepsilon_{xx} = M_0 + \int_{\kappa_0}^{\kappa} M_0\sigma_{xx,0} d\varepsilon_{xx}, \tag{3}
\]

where \( \kappa \in [\kappa_0, \kappa_{\text{p}}] \) depends on the yield condition. As long as the deformation in terms of \( \kappa \) is strictly increasing or decreasing, Eq. (3) constitutes a unique function \( M(\kappa) \).

4. Deformation history of the quasi-static strip model

In the quasi-static strip model, all cross sections experience the same deformation history. Hence, only the deformation history of one example cross section needs to be considered. At \( x_{BR} \), the initial values \( \kappa_{BR} \) and \( M_{BR} \) are defined by the deformation history in upstream process steps and at the bottom roll. Under normal process conditions, the direction of the deformation is monotonic between the closely spaced zinc bath rolls. Thus, the relation Eq. (3) between two rolls can be parameterized based on the strip curvatures at the previous rolls. For the complete domain, the bending moment \( M \) is thus given by

\[
x_{BR} < x < x_{CR} : M = M_{BR}(\kappa(x); [\kappa_{BR}, \kappa_{CR}]) \tag{4a}
\]

\[
x_{CR} < x < x_{SR} : M = M_{CR}(\kappa(x); [\kappa_{CR}, \kappa_{SR}]) \tag{4b}
\]

Due to the straightening effect of the tensile force \( N \) on the large domain \( x_{SR} < x < x_{TR} \), it is assumed that the curvature is purely elastically reduced to zero. \( M_{SR} \) can then be analytically calculated. The relations \( M_{BR}(\kappa; [\kappa_{BR}, \kappa_{CR}]) \) and \( M_{CR}(\kappa; [\kappa_{CR}, \kappa_{SR}]) \) can be efficiently numerically computed as a lookup table (LUT) before solving the BVP Eq. (1). Thus, the constitutive relations of the strip can be reused when the BVP is solved for different roll settings. Figure 3 shows these relations as piecewise defined models. The actual curvatures \( \kappa_{CR} \) and \( \kappa_{SR} \) are obtained together with the solution of the BVP Eq. (1) subject to Eq. (4).

5. Numerical implementation

The nonlinear BVP Eq. (1) is discretized by means of finite elements. For each element, the Galerkin weighted residual method is used, where Hermite polynomials of fifth order are employed as trial functions. As primary variables, the deflection, the slope and the curvature at the element nodes are chosen. This ensures continuity of these quantities. The algebraic problem is solved by means of the Newton-Raphson method. An additional contact algorithm determines the unknown points of contact such that the strip touches the rolls tangentially. For the computation of the stress profiles and the stress results, the cross section is discretized along the direction \( z \). At each grid point, the relations (2) are integrated by means of Runge-Kutta schemes.

References


Plastic Deformation of Axially Moving Continuum in Mixed Eulerian-Lagrangian Formulation

Peter G. Gruber¹, Yury Vetyukov², Michael Krommer¹

¹Linz Center of Mechatronics GmbH
Altenbergerstr. 69, 4040 Linz, Austria
e-mail: peter.gruber@lcm.at

²,³Institute of Mechanics and Mechatronics, Vienna University of Technology
Getreidemarkt 9, 1060 Vienna, Austria
e-mail: yury.vertyukov@tuwien.ac.at

Abstract

We present a new approach to model the motion of a metal sheet during a rolling mill process. In particular, we focus on the planar motion of the sheet as seen from the bird’s eye view in the area between two consecutive roll stands. The outflow of the sheet from the preceding roll stand, as well as the inflow of the sheet into the subsequent roll stand are subject to given entry and exit velocity profiles, which are inhomogeneous and varying in time. The method takes advantage of a mixed Eulerian-Lagrangian formulation, meaning that the governing equations are based on spatial coordinates and material points at the same time. The resulting Finite Element mesh is spatially fixed with respect to the direction of the rolling process, whereas material is transported across the geometry of the mesh. The overall deformation is described by a multiplicative scheme, allowing for an exact geometrical representation and the study of specifically non-linear effects. Particular emphasis is placed on modeling plastic material behavior, and the transport of the inelastic strain field required thereby.

Keywords: material generation, spatial description, finite element method, geometric nonlinearity, elastoplasticity, axially moving structures

1. Introduction

We investigate the numerical computation of the planar deformation of an axially moving strip with prescribed velocity distribution at spatially fixed interfaces. Such problems naturally occur in material forming processes as, e.g., the extrusion, rolling or coiling of thin strips of material. Analytical results on vibrations of axially moving one-dimensional structures are summarized in the review paper Ref. [3]. A Finite Element approach of this problem based on a Lagrangian formulation, however, would ultimately lead to meshes with materially fixed elements. Spatially fixed interface conditions would have to be imposed away from elements’ edges, and serious numerical troubles would arise, cf. the discussion of variational crimes in, e.g., Ref. [2].

In contrast to that, the presented approach utilizes a mixed Eulerian-Lagrangian formulation, leading to meshes which are spatially fixed in transport direction, whereas they move with the material in transverse direction. A consistent mathematical description of imposing the interface conditions at spatially fixed positions is thereby possible. We further focus on a consistent formulation of finite strain plasticity in the framework of the presented kinematic description, and we discuss the question of how to transport the plastic strain field in an optimal manner. The latter has been investigated lately by the authors of Ref. [5] for the case of one dimensional structures.

2. Kinematic Description

Let us consider a strip of continuous material, which is moving in the $(x, y)$-plane, see Fig. 1. The gross motion takes place in $x$-direction, such that at each time $t \in [0, T]$ a certain set of the strip’s material particles, called the active domain $\Omega(t)$, is located between the interfaces $\Gamma_{\text{entry}}$ at $x = 0$ and $\Gamma_{\text{exit}}$ at $x = L$.

![Figure 1: Material strip $\Omega(t)$ moving in between two spatially fixed interfaces $\Gamma_{\text{entry}}$ and $\Gamma_{\text{exit}}$.](image)

Material particles are considered to enter the active domain in form of strip segments with variable length $l(y, t) = v_{\text{entry}}(y, t) \, dt$ as outlined in the top left scene of Fig. 2. In the infinitesimal framework we may instead equivalently consider the application of an intrinsic deformation gradient $F$ applied to strip segments of constant length $l = v_e \, dt$, with $v_e$ denoting the strip’s nominal velocity. Thereby – and under consideration of additional plastic deformation $F_p$ – the active domain of the strip forms its reference configuration (see bottom left scene of Fig. 2) with $\hat{r}$ denoting the position of each material particle of the strip’s active domain. Elastic deformation $F_e$ finally leads to the actual configuration of the active domain (top right scene of Fig. 2), where

---

¹This work has been carried out at LCM GmbH as part of a K2 project. K2 projects are financed using funding from the Austrian COMET-K2 programme. The COMET K2 projects at LCM are supported by the Austrian federal government, the federal state of Upper Austria, the Johannes Kepler University and all of the scientific partners which form part of the K2-COMET Consortium.
The undeformed state of a domain is defined by the configuration \( r = \vec{r} + u_x \hat{i} + u_y \hat{j} \), where \( \hat{i} \) and \( \hat{j} \) are unit vectors in the \( x \)- and \( y \)-direction, respectively. The overall deformation gradient is given by

\[
F = F_e F_p F^*.
\]

In a classical Lagrangian formulation, material particles are identified by their position in the reference configuration \( \vec{r} \). In other words, the displacement solution \( u = (u_x, u_y) \) is obtained by integration of the total strain energy on the active domain in the reference configuration as a function of \( \vec{r} \). In contrast to that, we prevent the finite element mesh from traveling across the interface lines in axial direction by using a mixed Eulerian-Lagrangian formulation. Material particles are identified by their position vector \( \vec{r} = \vec{r} + u_x(\vec{r}) \hat{i} = \vec{r} - u_y(\vec{r}) \hat{j} \) in an intermediate configuration (bottom right scene of Fig. 2). Integration of the total strain energy is performed on the rectangular shape of the active domain in intermediate configuration, meaning, all field variables are considered to be functions of \( \vec{r} \).

### 3. Material model

Having accepted the multiplicative decomposition of the total deformation gradient in Eq. (2), we then turn to the application of a finite strain plasticity law. In this context we assume a rate dependent Mises solid with isotropic hardening. The presented mathematical model follows the monograph Ref. [1], the numerical solver is closely related to a method suggested by the authors of Ref. [4]. Numerical results, as those shown in Fig. 3, give rise to the question of proper strategies concerning the transport of the plastic and intrinsic deformation gradients \( F_p \) and \( F^* \).

### References


Complete modeling of the dynamics of sliding beams with large deformation

Alexander Humer\textsuperscript{1,3}, Loc Vu-Quoc\textsuperscript{2}, Ivo Steinbrecher\textsuperscript{3}

\textsuperscript{1,3}Institute of Technical Mechanics, Johannes Kepler University Linz
Altenberger Strae 69, 4040 Linz, Austria
e-mail: alexander.humer@jku.at, ivo.steinbrecher@jku.at

\textsuperscript{2}Department of Mechanical & Aerospace Engineering, University of Florida
231 MAE-A Building, Gainesville, FL 326250, USA
e-mail: vu-quoc@ufl.edu

Abstract

In the paper, a generalized formulation for the dynamic analysis of sliding beams undergoing large deformation is outlined. In contrast to previous contributions, which mainly focus on describing the beam behaviour within a certain spatially fixed domain, the proposed approach allows to take into account a complete sliding structure. For this purpose, the coordinate transformation typically employed in problems of this kind needs to be extended in order to account for the entire beam. If the individual parts of the beam are not decoupled by kinematic constraints, the transformation moreover depends on the structure current state of deformation in general. A simple numerical example is presented in order to show the potential of the proposed approach.

Keywords: sliding beam, axially moving continua, dynamics, large deformation, geometrically exact beam theory

1. Introduction

The paper discusses a formulation for the dynamic analysis of sliding beams that are deployed or retrieved through a prismatic joint. Sliding beams and the large class of problems subsumed under the term “axially moving continua” share several common characteristics. In either case, the structure under consideration typically undergoes a large rigid body motion into its axial direction that is superimposed by some more or less pronounced flexible deformation. As the terms “sliding” and “axially moving” already imply, the relative motion with respect to its support is a further typical feature of such structures. The importance of these closely related kinds of problems in industrial applications is reflected in the attention paid on it in the literature. Sliding or moving structures are integral parts in many engineering problems that range from telescopic antennas over belt drives to rolling mills, for example. The research on these topics dates back to the mid of last century [1] and is reviewed, e.g., in [2].

In the following, we study the dynamics of a sliding beam supported by a spatially fixed prismatic joint, see Fig. 1. The prismatic joint, which can be regarded as a guide, inhibits both a transverse deflection and a rotation of the beam cross-section, however, it does not constrain its axial motion. Being subjected to a large deformation superimposed on the sliding motion, the beam material points instantly located at the prismatic joint are not known in advance but depend on the structure current state of deformation. The prismatic joint divides the beam into two domains that generally change in the course of motion. Owing to the varying domains, the interface conditions naturally also change over time, which complicates a straight-forward solution by means of numerical methods requiring a spatial discretization.

The key idea of the proposed formulation is a deformation-dependent transformation of a beam material coordinate, which maps the two varying domains of the structure onto fixed ones. The simpler numerical treatment of the spatial problem enabled by such transformation, however, comes at the cost of a more complex structure of the inertia terms. As usual in the kinematics of relative motion, the material time derivatives of the displacement and the cross-sections angle of rotation are extended by stiffness and velocity convection terms. The paper combines and extends previous contributions on the topic of sliding beams [3, 4]. As opposed to [3], the formulation presented in [4] allows to consider the complete structure, i.e., the mutual influence of the beam parts inside and outside guide. Following [3], the geometrically exact theory for the planar motion of beams subjected to bending, stretching and shearing [5] is used in what follows, whereas [4] is restricted to slender structures, for which shear deformation is ignored.

2. Governing equations

We discuss the proposed formulation by means of a simple example problem, which can easily be extended to more complex situations. Consider a beam of length $L$ being retracted into a guide while undergoing large deformation. Here, we are not only interested in the beam deformation outside the guide, but also in its behaviour inside. Inside, the deformation is restricted to axial stretching due to the constraints at the opening of the guide, which prohibit a transverse deflection and a rotation of the cross-section.

Figure 1: A sliding beam being retrieved into an axial guide by a force $F$ applied at its left-hand tip

\cite{1} The author acknowledges the support by the Linz Center of Mechatronics (LCM) in the framework the Austrian COMET-K2 program.
The material point instantly located at the opening of the guide, however, changes in the course of motion, i.e., the corresponding kinematic constraints is prescribed at a non-material point. Let \( l_1(t), l_2(t) = L - l_1(t) \) denote the lengths of the (material) portions inside and outside the guide, respectively, we can partition the entire structure into two varying domains of the material axial coordinate \( X^1 = [0, L] \), i.e., \( X^1 \in [0, l_1(t)] \) inside and \( X^1 \in [l_1(t), L] \) outside, respectively. In order to circumvent varying domains and non-material constraint conditions, a stretched coordinate \( \xi^1 \) is introduced by means of a piecewise linear mapping of \( X^1 \) onto constant domains,

\[
\xi^1(X^1, t) = \begin{cases} \frac{1}{l_1(t)} l_1(0) + \frac{1}{l_2(t)} l_2(0) + \frac{1}{l_1(t)} l_1(t) & 0 < X^1 < l_1(t), \\ \frac{1}{l_1(t)} l_1(0) - \frac{1}{l_2(t)} l_2(0) + \frac{1}{l_1(t)} l_1(t) & l_1(t) < X^1 < L. \end{cases}
\]

These domains represent the (normalized) initial configuration concerning the beam parts inside and outside the guide,

\[
X^1 \in [0, l_1(t)] \rightarrow \xi^1 \in [0, l_1(0)/L], \quad X^1 \in [l_1(t), L] \rightarrow \xi^1 \in [l_1(0)/L, 1].
\]

Introducing such transformation, every material field quantity \( f \) has to be represented as function of the stretched coordinate \( \xi^1 \),

\[
f(X, t) = \tilde{f}(\xi^1(X, t), t) = \tilde{f}(\xi^1, t),
\]

which implies for the spatial and time derivatives that

\[
\frac{\partial f}{\partial X^1} = \frac{\partial \tilde{f}}{\partial \xi^1} \frac{\partial \xi^1}{\partial X^1},
\]

\[
\tilde{f} = \frac{\partial ^2 \tilde{f}}{\partial \xi^2}; \frac{\partial \tilde{f}}{\partial \xi}; \frac{\partial \tilde{f}}{\partial t^2}; \frac{\partial \tilde{f}}{\partial \xi^2}; \frac{\partial \tilde{f}}{\partial t}; \frac{\partial \tilde{f}}{\partial \xi}; \frac{\partial \tilde{f}}{\partial t^2} + \left( \frac{\partial \xi^1}{\partial t} \right)^2 \left( \frac{\partial \xi^1}{\partial t} \right)^2.
\]

On one hand, the transformation allows to prescribe the constraints for the transverse deflection \( \tilde{u}^2 \) and the angle of rotation \( \tilde{\psi} \) at the opening of the guide at fixed points with respect to \( \xi^1 \),

\[
u_0^2(l_1(t), t) = \tilde{u}^2_0(l_1(0)/L, t) = 0, \quad \psi_0(l_1(t), t) = \tilde{\psi}^2_0(l_1(0)/L, t) = 0.
\]

On the other hand, Eq. (4) already suggests the more complicated structure of the inertia terms in the equations of motion. Indeed, a Galerkin projection of the equations of motion in weak form, i.e., d’Alembert’s principle, gives semi-discrete equations of motion for the vector of generalized degrees of freedom \( \mathbf{d} \),

\[
\mathbf{M}(l_1(t))\ddot{\mathbf{d}} + \mathbf{V}(l_1(t))\dot{\mathbf{d}} + \mathbf{S}(l_1(t))\mathbf{d} + \mathbf{f}(\mathbf{d}, l_1(t)) = \mathbf{F}(t).
\]

where terms related to velocity convection \( \mathbf{V} \) and stiffness convection \( \mathbf{S} \) enter the equations of motion in addition to a state-dependent mass matrix \( \mathbf{M} \). The non-linear internal forces are represented by \( \mathbf{F} \); the external forces are collected in \( \mathbf{F} \). As opposed to common notion, the velocity convection matrix \( \mathbf{V} \) is not skew-symmetric in general, see [4] for discussion.

In most investigations on sliding beams and axially moving continua, the motion of the structure is prescribed kinematically at the boundaries of the varying domains, i.e., the opening of the guide in the present case. If we abandon such assumption, which inertially decouples the individual domains, the (material) length of portion inside guide \( l_1(t) \) is no longer known in advance but depends on the current state of deformation. For the present problem, it is understood that \( l_1(t) \) then equals the negative axial displacement of the material point currently located at the opening,

\[
l_1(t) = -\tilde{u}^2_0(l_1(0)/L, t).
\]

Consequently, the matrices in Eq. (7) are not only time-dependent, but we have to deal with non-linear inertia terms in such a situation.

3. Numerical example

We apply the proposed formulation to a familiar numerical example, the sliding spaghetti problem. In contrast to the classical spaghetti problem, the sliding motion is not prescribed kinematically at the opening of the guide. Instead, it originates from an axial force \( F = 250 \text{N} \) applied at the beam’s left-hand tip, see Fig. 1. The beam is \( L = 6 \text{ m} \) long, where the part initially stored in the guide is \( l_1(0) = 1 \text{ m} \). Its rectangular cross-section is \( 0.2 \text{ m} \) high and \( 1 \text{ m} \) wide; the density is \( \rho = 10^3 \text{ kg/m}^3 \). The elastic properties are chosen such that the effective bending, shear and extensional stiffnesses are \( EA = 2 \times 10^6 \text{ N} \), \( GA = 6.54 	imes 10^5 \text{ N} \) and \( EI = 6.67 \times 10^7 \text{ Nm}^2 \).

We investigate a flexible beam that is subjected to large deformation under its own weight. At \( t = 0 \), the structure is released from an undeformed straight configuration at rest.

![Figure 2: Sequence of configurations of the beam axis](image)

A first idea of the developed dynamic behaviour of the beam during the retraction is illustrated in Fig. 2, in which a sequence of snapshots of its axis is depicted. Despite gravity, the beam is deflected in the opposite direction due to its sliding motion. Due to the limited space, we refer to future publications for an in-depth investigation on the dynamics of sliding beams, in which, e.g., aspects of stability and energy distribution are examined.

References

Formulating the perfectly matched layer as a control optimization problem

Daniel Ritzberger\(^1\), Alexander Schirrer\(^2\), Stefan Jakubek\(^3\)

\(^1,^2,^3\) Vienna University of Technology, Austria
e-mail: daniel.ritzberger@gmail.com\(^1\)

Abstract

An automated approach to emulate the absorbing properties of a perfectly matched layer (PML) in wave equations is presented. Instead of applying the coordinate stretching to obtain a modified PML wave equation, a feedback boundary controller is parameterized. The set of unknown control parameters is obtained through genetic optimization by minimizing the error between the wave equation with additional feedback controller and the desired damped fundamental solution at certain frequency pairs. With this approach the time-consuming task of constructing a PML, especially for complex wave-like equations like the moving Euler-Bernoulli beam, is automated and it leads to an easy-to-implement and computationally efficient alternative.

Keywords: Absorbing boundary conditions, Genetic optimization, Euler-Bernoulli beam

1. Introduction

In many applications where an unbounded solution of a wave-like equation is desired, the problem occurs that due to limited computational capabilities the domain has to be truncated at some point. To let the solution of this confined domain approximate the free-wave propagation, boundary conditions with absorbing properties have to be applied. The work by Engquist and Majda [1] addressed this issue and absorbing boundary conditions (ABCs) were derived which worked well under certain circumstances. The technique to surround the computational domain with a perfectly matched layer was first described by Berenger [2] for the absorption of electromagnetic waves. The idea of the perfectly matched layer was later extended and applied to other wave propagation problems, both in a split or un-split field formulation [3] [4].

The key idea of the perfectly matched layer is that if the fundamental solution of a wave equation is evaluated along a complex coordinate an additional damping is gained. This can be easily shown investigating the one-dimensional wave equation

\[
\frac{\partial^2 w(x,t)}{\partial t^2} = c^2 \frac{\partial^2 w(x,t)}{\partial x^2} \tag{1}
\]

with its fundamental solution

\[
w(x,t) = e^{i\omega_x x} e^{i\omega_t t} \tag{2}
\]

where \(\omega_x\) is the so called wavenumber or spatial frequency and \(\omega_t\) the angular frequency. If this fundamental solution is evaluated along a contour that is stretched into the complex plane \(\tilde{x} = x + i f(x)\), Eq. (2) can be rewritten as

\[w(\tilde{x},t) = e^{i\omega_x (x+i f(x))} e^{i\omega_t t} = e^{-\omega_x f(x)} \underbrace{e^{i\omega_x x} e^{i\omega_t t}}_{w(x,t)} \tag{3}
\]

Note, that when \(f(x)\) is zero the original fundamental solution is obtained, whereas if \(f(x) > 0\) an exponential decay is added. The wave equation with respect to its complex coordinate is then transformed back to its real-valued coordinate using

\[
\frac{\partial \tilde{w}}{\partial \tilde{x}} = (1 + i \frac{df(x)}{dx}) \frac{\partial w}{\partial x} \quad \frac{\partial \tilde{w}}{\partial \tilde{t}} = \frac{\partial w}{\partial t} + \frac{i}{1 + i \frac{df(x)}{dx}} \frac{\partial w}{\partial x} \tag{4}
\]

This transformation is a tedious task especially when spatial derivatives of higher orders are involved as, for example, in the Euler-Bernoulli beam equation and usually involves using several auxiliary variables which increases the computational effort.

2. PML as a control optimization problem

For demonstration purposes, again the scalar wave equation (1) is considered for deriving the PML as a control optimization problem. As it is shown later, this method can easily be adapted for controlling different, more complex, wave-like equations such as the Euler-Bernoulli beam equation. Discretizing the scalar wave equation using central finite difference approximations on a uniform grid results in

\[w^{j+1}_n - 2w^j_n + w^{j-1}_n = c^2 \frac{\Delta t^2}{\Delta x^2} (w^{j+1}_{n+1} - 2w^j_n + w^{j-1}_{n-1}) \tag{5}
\]

which can be aggregated for every node into a discrete state-space system

\[
x^{j+1} = Ax^j \tag{6}
\]

where \(x^j = [w^n, w^{j-1}]^T\) is the solution vector at the discrete time \(j\Delta t\). The fundamental solution becomes

\[w_n^j = e^{i\omega_x \Delta x n} e^{i\omega_t \Delta t j} \tag{7}
\]

Inserting (7) into the discretized wave equation (5) results in the so-called dispersion relation which expresses the dependency between \(\omega_x\) and \(\omega_t\). There exist infinitely many \(\{\omega_x, \omega_t\}\)-pairs but the magnitudes of \(\omega_x \Delta x \) and \(\omega_t \Delta t\) can be confined between \([-\pi, +\pi]\). Higher magnitudes can not be resolved by the grid. To control the system so that it has a reflection-less exponential decay of the solution inside a layer surrounding the computational domain, a state feedback controller is added.

\[x^{j+1} = Ax^j + BKx^j \tag{8}
\]

where the control matrix \(K\) is defined to have diagonal substructures of the form

\[K = \begin{pmatrix} k_1 & 0 & \cdots & & \cdots & \cdots \\ k_2 & k_{p+1} & \cdots & & \cdots & \cdots \\ & & k_{p+2} & \cdots & \cdots & \cdots \\ & & & k_p & \cdots & \cdots \\ & & & & k_{2p} & \cdots \\ & & & & & k_{2p} & \cdots \\ \end{pmatrix} \tag{9}
\]
where \( p \) is the number of nodes that are inside the damping layer. The input matrix \( B \) distributes the control input to the corresponding nodes of the damping layer.

The desired behavior can be analytically given for a fundamental wave, e.g. a single frequency pair \( (\omega_i, \omega_f) \), by evaluating the discrete form of (3). For \( f(x) \), a function that is zero inside the computational domain, increasing with second or third order within the damping layer and continuous at the interface is preferable. Let this desired fundamental solution be denoted as \( \omega_{fund}(\omega, \omega_x, t) \), where \( \omega \in \Omega \) is a certain frequency pair and \( \Omega \) a set containing a finite number of pairs.

The eq. (6) is initialized with \( \omega^0 = [\omega_{fund}, \omega_{fund}] \) and continued for certain amount of time steps \( j_{max} \). The error between the fundamental solution and the controlled state space system is aggregated over time and the frequency set \( \Omega \) to form the objective function

\[
J(K) = \sum_{\Omega} \sum_{j=2}^{j_{max}} |\omega^j_{fund}(\omega, \omega_x, t) - w^j(K)|^2 \tag{10}
\]

The objective function is then minimized using a genetic algorithm to obtain the optimal control matrix \( K \). To address stability of the controlled damping layer, the eigenvalues of the state space system (8) are evaluated during the optimization and destabilizing controllers are penalized with a multiplicative weighting term.

3. Resulting controller for moving Euler-Bernoulli beam

The procedure described above is used to find a state feedback controller that emulates PML properties on one side for the moving Euler-Bernoulli beam equation

\[
pA\dddot{w} = -EI\dddot{w} + (T - pA^2)v'' + 2wpA\dot{w}' \tag{11}
\]

where \( pA \) is the mass per unit length, \( EI \) the bending stiffness, \( T \) the tensile force and \( v \) the speed of the moving coordinate. Figure 1 shows the normalized phase velocity for the non-moving and the moving Euler-Bernoulli beam. Substantial dispersion occurs due to the bending stiffness. Furthermore, the two branches for left and right going waves are not symmetrical for the moving Euler-Bernoulli beam.

![Figure 1: Normalized phase velocity for left and right going waves over the spatial frequency. The phase velocity for the non-moving EBB (black) is symmetric around zero whereas the branches for the moving EBB (red) are tilted.](image)

The parameters used are shown in Table 1. When discretizing to obtain the state space system the spatial grid size was set to \( \Delta x = 0.4 \) [m] and the temporal grid size to \( \Delta t = 7 \times 10^{-4} \) [s].

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>mass per unit length</td>
<td>( \rho A )</td>
<td>1.35 [kg/m]</td>
</tr>
<tr>
<td>bending stiffness (contact)</td>
<td>( EI )</td>
<td>150 [Nm²]</td>
</tr>
<tr>
<td>tensile force (contact)</td>
<td>( T )</td>
<td>20 [kN]</td>
</tr>
<tr>
<td>speed</td>
<td>( v )</td>
<td>50 [m/s]</td>
</tr>
</tbody>
</table>

The state feedback controller actuates \( p = 10 \) nodes. For the two outer nodes of the damping layer Dirichlet boundary conditions are applied. The set \( \Omega \) consists of 10 pairs where \( \omega_i \Delta t \) is equidistantly spaced between 0 and \( \pi \) and the corresponding \( \omega_f \Delta t \) are calculated from the dispersion relation. The maximum number of time steps for which the objective function is evaluated is set to \( j_{max} = 100 \). The satisfactory performance of the optimized feedback controller is illustrated in Figure 2. No significant reflections into the domain are produced.

![Figure 2: The optimized feedback controller is applied on the left boundary. A frequency sweep excites the right boundary. No reflections back into the computational domain are visible.](image)

4. Conclusion

In the work the construction of a PML is described as an optimization problem to obtain a feedback controller. The method is applied for the moving Euler-Bernoulli beam and it was shown in numerical results that a high absorption is achieved. The procedure is highly automated, and the mathematical effort for the user is reduced to determining the dispersion relation instead of performing the original PML transformation which is a task especially tedious with wave-like equations of high orders of (mixed) derivatives.

References

Vibrations of a double-beam complex system subjected to a moving force

Jarosław Rusin

Faculty of Civil Engineering, Architecture and Environmental Engineering, University of Zielona Góra
prof. Z. Szafrana 1, 65-516 Zielona Góra, Poland
e-mail: J.Rusin@ib.uzgora.pl

Abstract

In this paper the dynamic response of a complex double-beam system traversed by moving load is considered. The conflation is represented by a set of linear springs in Winkler model. The classical solution of the response of complex systems subjected to a force moving with a constant velocity has a form of an infinite series. The main goal of this paper is to show that in the considered case part of the solution can be presented in a closed, analytical form instead of an infinite series. The presented method to search for a solution in a closed, analytical form is based on the observation that the solution of the system of partial differential equations in the form of an infinite series is also a solution of an appropriate system of ordinary differential equations. The dynamic influence lines of complex systems may be used for the analysis the complex models of moving load.

Keywords: vibration complex systems, moving loads, closed solutions

1. Introduction

Composite beams are very important in aeronautical, civil and mechanical engineering as structural members with high strength to weight ratios. Dynamics response of complex beam systems has been studied by many authors in the recent decades. The problem of a dynamic response of a structure subjected to moving loads is interesting and important. This problem occurs in dynamics of bridges, aircrafts, missiles as well as railways and motorways. The problem of vibration of the complex beams and strings has been considered in the papers [1,2,3,4,5,6]. It would be interesting to study the problem of dynamic response of a composite beam to moving loads. The paper includes the study of a dynamic response of a finite, simply supported double-beam complex system subject to a force moving with a constant velocity. The classical solution has a form of an infinite series. The main goal of the paper is to show that the aperiodic part of the solution can be presented in a closed form instead of an infinite series. In the paper the solutions are derived in a closed form Ref. [2, 5, 6]. The presented method to finding a solution in a closed form is based on the observation that the solution of the system of partial differential equations in the form of an infinite series is also a solution of an appropriate system of ordinary differential equations Ref. [4,5]. The solution for the dynamic response of the composite beam under moving force is important because it can be used also in order to find the solution for other types of moving loads. The double beam connected in parallel by a linear elastic elements can be studied as a theoretical model of composite beam in which bending and coupling effects are taken into account.

2. Mathematical model and governing equation

Let us consider the vibration of complex system consist of couple Euler-Bernoulli beams interface by a linear springs in Winkler model under axial compression $N_i$ and $N_c$ excited by a force $p(x,t)$ moving with a constant velocity $v$ as on Fig.1. The solution of the system in the classical forms is investigated it is possible to find the closed forms of the deflection function and internal forces.

$$p(x,t) = p(x,t)$$

$$N_i, N_c$$

$$EI, m_i, m_c$$

$$w_i(x,t), w_c(x,t)$$

Figure 1: Double-beam system under a moving force

The transverse vibration of double-beam system is governed by two conjugate partial differential equations

$$EI \frac{\partial^4 w_i}{\partial x^4}(x,t) + N_i \frac{\partial^2 w_i}{\partial x^2}(x,t) + m_i \ddot{w}_i(x,t) + \frac{k}{2} \left[ w_i(x,t) - w_c(x,t) \right] = p(x,t)$$

$$EI \frac{\partial^4 w_c}{\partial x^4}(x,t) + N_c \frac{\partial^2 w_c}{\partial x^2}(x,t) + m_c \ddot{w}_c(x,t) + \frac{k}{2} \left[ w_i(x,t) - w_c(x,t) \right] = 0$$

where $E$ is the flexural rigidity of the beams, $I$ is Young’s modulus of elasticity, $I$ is the moment of inertia of the cross-section area of the beam, $m$ is the mass spread over the length of the $i$-th beam and $k$ is the stiffness modulus of a Winkler elastic element. The load function in Eqn (1) and from Fig. 1 has the form

$$p(x,t) = P \delta(x - vt)$$

where $\delta(\cdot)$ is the Dirac delta. After introducing the dimensionless variables

$$\xi = x/L, \quad T = vt/L, \quad \xi \in [0,1], \quad T \in [0,1]$$

the differential equations of motion of the beams system have the form

$$w_i^{\prime\prime\prime}(\xi,T) + N_{i1} w_i^{\prime\prime}(\xi,T) + \sigma_i \dot{w}_i(\xi,T) + \frac{k}{2} \left[ w_i(\xi,T) - w_c(\xi,T) \right] = P_1 \delta(\xi - T)$$

$$w_c^{\prime\prime\prime}(\xi,T) + N_{c1} w_c^{\prime\prime}(\xi,T) + \sigma_c \dot{w}_c(\xi,T) + \frac{k}{2} \left[ w_i(\xi,T) - w_c(\xi,T) \right] = 0$$

The parameters from Eqn (5) and Eqn (6) have the following
designations
\[ N_{ii} = \frac{N_i L^2}{EI_i}, \sigma_i^2 = \frac{m_i v_i^2 L^2}{EI_i} = \frac{v_i^2}{v_i}, k_i = \frac{k_i L_i}{EI_i}, P_i = \frac{P_i L_i}{EI_i} \]  
(7)

where \( v_i \) is the shear wave velocity propagated in the first and second beam respectively. Therefore, boundary conditions Eqn (5) and Eqn (6) have the form
\[ w_i(0, T) = w_i(1, T) = 0, \quad w_i'(0, T) = w_i'(1, T) = 0 \]  
(8)

Let the initial conditions have the form
\[ w_i(\xi, 0) = 0, \quad w_i'(\xi, 0) = 0 \]  
(9)

The solutions of Eqn (5) and Eqn (6) for boundary conditions (8) are assumed to be in the form of sine series
\[ w_i(\xi, T) = \sum_{n=1}^{\infty} v_i n \sin n \pi \xi \]  
(10)

Eventually, the solution of the system Eqn (5) and (6) are sums of the particular integrals \( w_i(\xi, T) \) and general integrals \( w_i(\xi, T) \) take the form
\[ w_i(\xi, T) = w_i^p(\xi, T) + w_i^s(\xi, T) \]  
(11)

3. The classical and closed form solutions

Classical part of the particular solution presents itself as
\[ w_i^p(\xi, T) = \sum_{n=1}^{\infty} P_i \left( k_i - b_i \sigma_i^2 \right) \sin n \pi \xi \sin n \pi \xi \]  
(12)

\[ w_i^s(\xi, T) = \sum_{n=1}^{\infty} a_i \left( a_i \sigma_i + a_i \sigma_i^2 \right) \sin n \pi \xi \sin n \pi \xi \]  
(13)

where
\[ a_i = k_i \left( N_i + \sigma_i \right) + k_i \left( N_i + \sigma_i \right), \quad a_i = N_i + N_i + \sigma_i + \sigma_i^2 \]  
(14)

Let us notice that these functions are solution not only to the system of partial differential system Eqn (5) and (6) but also to the system of ordinary system equations Ref. [4, 5, 6]
\[ w_i^p(\xi, T) + a_i w_i^p(\xi, T) + a_i w_i^p(\xi, T) + a_i w_i^p(\xi, T) = P_i \left( k_i \delta(\xi - T) + b_i(\xi - T) + \delta(\xi - T) \right) \]  
(15)

\[ w_i^s(\xi, T) + a_i w_i^s(\xi, T) + a_i w_i^s(\xi, T) + a_i w_i^s(\xi, T) = P_i k_i \delta(\xi - T) \]  
(16)

which conforms with the following boundary conditions
\[ w_i^p(0, T) = w_i^s(1, T) = 0, \quad D = \{1, \overline{1}, IV, IV, VII\} \]  
(17)

where \( D \) is order derivative. Eqn (15) and (16) lead to the analytical closed-form.

4. Numerical results

The following dimensionless values of the parameters are used in the numerical calculations: \( N_1 = 2, N_2 = 3, \sigma_1 = 0.1, \sigma_2 = 0.2, k_1 = 20, k_2 = 10, P_1 = 20 \). The results for different location of the moving point force are presented in graphical form in Fig. 2. The continuous line represents the deflections of the loaded beam. The dashed line shows deflection functions of the second beam for which the load is transferred with the coupling.

5. Conclusion

The motion of the system is described by a non-homogeneous conjugate set of two partial differential equations and aperiodic vibrations can be described by ordinary system equations. The classical solution for transverse displacement function has a form of a sum of two infinite series. It has been shown that aperiodic vibrations of the beam of the series can be presented in a closed, analytical form. The closed solutions take different forms depending if the velocity \( v_0 \) of a moving force is smaller or bigger than the shear wave velocity \( v_0 \) of the beams. The closed solutions improve the preciseness of the classical sine series expansion of the complex system of beams response by considering the aperiodic part as the solution not only to partial of the differential equation but also to appropriate ordinary differential equation. Having determined the closed form allows us to assess a quantity of necessary function approximation to determine the classical solution.

References


Online Parameter Identification for Traffic Simulation via Eulerian and Lagrangian Sensing

Elvira Thonhofer¹, Stefan Jakubek²

¹,²Institute of Mechanics and Mechatronics, Vienna University of Technology
Getreidemarkt 9/E325, A-1060 Vienna, Austria
e-mail: elvira.thonhofer@tuwien.ac.at¹, stefan.jakubek@tuwien.ac.at²

Abstract

The paper deals with the parameter identification for traffic models. Two fundamentally different approaches are presented, which are combined to the best advantage. The Eulerian sensing results in the sensors spatially stationary, i.e. fixed in place, while the Lagrangian sensing makes the sensors move with the traffic flow as sensor-vehicles. We present a method to identify model parameters from both methods individually and investigate identifiability via the Fisher Information Matrix. Additionally an approach to combine both methods is presented, particularly important while the data quality of a single method is not sufficient to identify all model parameters. Results for both Eulerian and Lagrangian sensing as well as the combined method are presented.

Keywords: Lagrangian sensing, optimization, Fisher information, identification

1. Introduction

Large scale traffic simulations are based on a macroscopic traffic model, where the vehicle flow is described by the transport equation, regarding traffic like a compressible fluid. The relationship between the speed of the flow and the local traffic density is described by a fundamental diagram (FD) [1]. We use a first order numerical scheme that works with an arbitrary, piecewise differentiable FD [3]. In order to identify the traffic model parameters, e.g. the parameters of the FD, we formulate an optimization problem based on a traffic model and data collected from sensors. Traditionally, stationary loop sensors are used to collect measurement data, representing the Eulerian sensing (ES) approach. The main drawback of this method is expensive installation, especially for urban networks, where significantly more sensors are required compared to highway applications. Hence, Lagrangian sensing (LS) via mobile sensors moving with the flow is an active research topic. Both specially equipped vehicles, so-called floating cars, and smartphones are in use.

The aforementioned macroscopic traffic model is used in combination with ES, and a simple microscopic model in combination with LS to keep the computational costs low. At this point the quality of the collected data is critical. Using the Fisher Information Matrix (FIM) for ES optimum stationary sensor placement for a range of realistic traffic situations can be determined. Additionally, parameters not possible to be identified via ES alone may be identified via LS to yield a sufficiently accurate set of model parameters for the macroscopic traffic model.

2. Macroscopic Traffic Model

The macroscopic description of traffic is based on the transport equation, a hyperbolic partial differential equation

$$\frac{\partial q(x,t)}{\partial t} + \phi(q(x,t)) \frac{\partial}{\partial x} = 0, \quad x \in \mathbb{R}, t > 0 \quad (1)$$

where $q$ represents the vehicle density in [veh./m], $x$ and $t$ represent the spatial and temporal coordinates respectively, and $\phi(q)$ represents the flux function

$$\phi(x,t,q) = f(q) = q(x,t) \cdot v(q, \theta), \quad [\text{veh./s}] \quad (2)$$

based on the (arbitrary) FD $v(q)$. The vector $\theta$ collects the parameters of the FD.

In order to solve Eq. (1) the spatial domain is discretized into cells along the length of a given road section. The vehicle balance for each cell $i$ at a discrete time $n$ is given by

$$\hat{q}_i^{n+1} = \hat{q}_i^n + (\phi_i^n - \phi_i^0) \frac{\Delta t}{\Delta x}, \quad i \in \mathcal{D}, n \geq 0 \quad (3)$$

where $\phi_i^n$ denotes the flux entering cell $i$ from the left, $\phi_i^n$ denotes the flux exiting cell $i$ and $\mathcal{D}$ denotes the discrete spatial domain. The fraction $\frac{\Delta t}{\Delta x}$ accounts for temporal and spatial increments, restricted by the Courant-Friedrichs-Lewy (CFL) condition for convergence. Meeting the CFL condition ensures that the numerical domain of dependence includes the physical domain of dependence.

3. Microscopic Traffic Model

An auto-regressive car-following model where a floating car follows a reference vehicle is used. The position $\hat{x}$ of the floating car at time step $n+1$ is updated according to the model equation

$$\hat{x}^{n+1} = \hat{x}^n + v(h, \theta) \cdot \Delta t \quad (4)$$

where $v(h, \theta)$ denotes the current velocity, which depends on the local traffic density represented by the measured headway $h$ with respect to the reference vehicle at $x_{ref}$ and the parameters $\theta$ of the FD.

4. Parameter Identification

The set of model parameters $\theta$ is identified by means of optimization. The model with its boundary conditions (BC) and initial conditions represents the system. The optimization goal is to minimize the error $J_{OE}$ computed by the objective function for a given system by variation of the decision variables $\theta$ while observing constraints on $\theta$. For ES, the objective function evaluates the normalized output error defined by

$$J_{OE,ES} = \sum_{s,n} \frac{\left( \hat{q}_{s,n} - q_{s,n}^{ref} \right)^2}{\max(q_{s,n})} \quad (5)$$
where the sensor locations are denoted by $s$ and the measurement data from a traffic sensor at a location $(j, s)$ is denoted $q_{j,s}^n$.

Equivalently, for the LS the cumulative prediction error of the model is defined by

$$J_{OE,LS} = \sum_{j,n} (\hat{x}_{j}^n - x_{j}^n)^2$$

(6)

where $x_{j}^n$ represents the true position of the floating car $j$ at a time step $n$.

4.1. Parameter Sensitivity

The parameter sensitivity vector $\psi_{i,OE}$ is defined by the total derivative of the output with respect to $\theta$,

$$\psi_{i,OE,ES}^n = \frac{\partial \hat{q}_{i}^n(\theta)}{\partial \theta} + \frac{\partial \hat{q}_{i}^n(\theta)}{\partial x^n(\theta)} \frac{dx^n(\theta)}{d\theta}$$

(7)

with $\hat{q}_{i}^n = f(q_{i-1}^{n-1}, \hat{q}_{i-1}^{n-1}, \theta)$ as defined by Eq. (3), and $x^n(\theta)$ denoting the regressor vector containing the corresponding past model outputs $x^n(\theta) = [\hat{q}_{i-1}^{n-1}, \hat{q}_{i-1}^{n-1}]^T$.

For LS and a vehicle $j$, Eqn (7) and the chain rule yield

$$\psi_{j,OE,LS}^n = \frac{\partial \hat{x}_{j}^n(q, \theta)}{\partial \theta} = \frac{\partial \hat{x}_{j}^{n-1}(\theta)}{\partial \theta} + \Delta t \frac{\partial \hat{q}_{j}^{n-1}(q(\theta), \theta)}{\partial \theta}$$

(8)

4.2. Identifiability

In order to assess identifiability of parameters we utilize the FIM in the output error (OE) configuration [2] for both ES and LS defined by

$$I_{OE} = \frac{1}{\sigma^2} \sum_{i,n} (\psi_{i,OE}^n)^T (\psi_{i,OE}^n)$$

(9)

where $\sigma^2$ is the (minimum) standard deviation of the estimated parameters.

We use a scalar criterion $J_S = \sigma_{min}(I)$ for evaluation where the smallest singular value $\sigma_i$ of $I$ is maximized. A singular value $\sigma_i = 0$ of $I$ indicates a non-influential (hence, non-identifiable) right-singular vector $v_i$, which in turn corresponds to non-identifiable parameter(s) of $\theta$.

5. Mixed sensing

Ill-conditioned singular values indicate that some parameters cannot be estimated well from the data set at hand, hence, the data quality is insufficient. However, parameters that can be identified from a certain measurement data set still may be used. Applying both the estimated parameters based on Lagrangian and Eulerian sensing allows for high accuracy at the same time keeping the sensor-costs low. Blending of the parameter sets is performed by weights $w_{ES,LS} \in [0, 1]$, which depend on the corresponding (relative) $\sigma_i$ and $v_i$. Note that blending algorithms must take spatial and temporal relevance into account.

The example illustrates insufficient Eulerian data, complemented by data of one floating vehicle, such that all parameters $\theta$ can be estimated reliably.

6. Example

We consider a single-lane road with a total length of 2500m and a speed limit of 13.89m/s (50km/h). The spatial increment $\Delta x = 50m$, the temporal increment is $\Delta t = 1s$, with a total simulation time of $t_{sim} = 600s$. The road is initially empty, vehicles enter at the left boundary and stop at the right boundary (red light). Results are validated via simulation with VISSIM®.

We use a single sensor location at position $x = 2400m$, recording density values at every time step during the simulation.

The collected data set is used in Eq. (5). Evaluating Eq. (9) shows that all parameters cannot be identified properly in detail parameters corresponding to the high-speed-low-density part of the FD are affected. Parameter identification through optimization itself is done by genetic optimization.

Application of one additional floating car from within the first vehicle platoon provides data to reliably identify the high-speed-low-density parameters, while, the data from this floating car is insufficient to identify the remaining parameters of the FD. Combining both sets of parameters weighted by $\sigma_i$ only, since both spatial and temporal relevance is given in this example yields a reliable parameter vector. Spatial and temporal relevance depend strongly on weather conditions and traffic incidents.

The resulting optimum parameter vector $\theta_{opt}$ is validated via simulation. The results together with the reference solution generated by VISSIM® at various time steps are depicted in Fig. 1.

7. Conclusion

The paper briefly introduces Eulerian and Lagrangian sensing and their relevant application in parameter identification for traffic simulations. Parameter sensitivity and identifiability are investigated and a blending method is proposed. Results are validated by means of simulation, however, real measurement data is currently collected for a rigorous validation. The LS data become readily accessible as vehicles are equipped with ACC and navigation systems and a Car2Infrastructure communication becomes standard equipment.

References


Modeling finite deformations of an axially moving elastic plate with a mixed Eulerian-Lagrangian kinematic description

Yury Vetyukov¹, Peter G. Gruber², Michael Krommer¹∗
¹, ³Institute of Mechanics and Mechatronics, Vienna University of Technology
Getreidemarkt 9, 1060 Vienna, Austria
e-mail: yury.vetyukov@tuwien.ac.at¹, michael.krommer@tuwien.ac.at³
²Linz Center of Mechatronics GmbH
Altenberger Straße 69, 4040 Linz, Austria
e-mail: peter.gruber@lcm.at

Abstract

We consider the motion of a flexible plate across a domain, bounded by two parallel lines. Kinematically prescribed velocities of the plate, entering the domain and leaving it, may vary in space and time. The corresponding deformation of the plate is quasistatically analyzed using the geometrically nonlinear model of a Kirchhoff shell with a mixed Eulerian-Lagrangian kinematic description. In contrast to the formulations, available in the literature, both the in-plane and the out-of-plane deformations are unknown a priori and may be arbitrarily large. The particles of the plate travel across a finite element mesh, which remains fixed in the axial direction. The evident advantage of the approach is that the boundary conditions need to be applied at fixed edges of the finite elements. In the paper, we present the mathematical formulation and demonstrate its consistency by comparing the solution of a benchmark problem against results, obtained with conventional Lagrangian finite elements.

Keywords: Axially moving plates, nonlinear theory of shells, multiplicative decomposition, Eulerian-Lagrangian description, finite element method

1. Introduction

The problem of mathematical modeling of nonlinear deformations of axially moving structures is both challenging and practically important. Numerous papers deal with the transverse vibrations of axially moving beams and strings, see a review paper Ref. [1]. While an extension towards nonlinearly coupled in-plane and out-of-plane vibrations of a moving plate is presented in Ref. [4], this model is incapable of representing arbitrarily deformed configurations of the plate. Moreover, the use of Lagrange equations of motion to an open system with influx and outflux of the mass is not justified in the latter reference.

Large axial deformation and bending of a beam, which can move across a fixed domain, is treated by the authors of Ref. [5] using a suitable change of variables. We apply a similar technique for the quasistatic modeling of finite deformations of a plate moving across a given domain in the direction x. The velocities of the plate are prescribed at two boundaries of the domain x = 0 and x = L, see Fig. 1.

![Figure 1: Quasistatic deformation of a plate with prescribed velocities at the boundaries](image1)

2. Mathematical model

In the present study we assume the velocity \( v_{\text{entry}} \), with which the plate is entering the domain, to be constant. In the future, arbitrary velocity profiles may be incorporated into the model using the notion of intrinsic strains and the technique of multiplicative decomposition of the deformation gradient, Ref. [6]. The varying velocity profile \( v_{\text{exit}}(y) \), with which the material particles of the plate are leaving the domain at \( x = L \), leads to the time varying deformation of the plate. Searching for a sequence of quasistatic equilibrium states of the elastic plate, we need to minimize the total energy of the active region of the plate, which is currently residing in the considered domain. Not going into details concerning the time integration, which is intended to be discussed in future publications, we focus on the kinematic modeling of the deformation of the plate.

![Figure 2: Two-stage mapping from the reference configuration to the actual one: the intermediate configuration is fixed in space](image2)
The plane reference configuration \(0 \leq \hat{y} \leq w\) is straight \((w\) is the undeformed width and \(\hat{r} = \hat{\mathbf{x}} + \hat{y} \hat{n}\) is the position vector in the reference configuration), see Fig. 2. The present mixed Eulerian-Lagrangian kinematic description makes use of a fixed intermediate configuration with the position vector \(\hat{r}\) such that the mapping of the positions of particles from the reference configuration to the actual one \(\mathbf{r} = \mathbf{r}(\hat{F})\) comprises two stages:

\[
\mathbf{r} = \hat{\mathbf{r}} + u_x(\hat{r}) \hat{x} + u_y(\hat{r}) \hat{y}, \quad \hat{\mathbf{r}} = \hat{\mathbf{r}} + u_z(\hat{r}) \hat{z}.
\]

(1)

The simplicity of this description essentially distinguishes it from the known Arbitrary Lagrangian-Eulerian formulation, Ref. [2]. All fields are functions of the place in the fixed intermediate configuration, in which a finite element discretization is performed.

We apply the classical Kirchhoff shell model, see Refs. [3, 7]. The total gradient of deformation of the plate with the differential operator of the intermediate configuration \(\nabla\) results in the form

\[
\mathbf{F} = \hat{\nabla} \mathbf{r} = \hat{\nabla} \mathbf{r}^T \cdot \hat{F}, \quad \hat{\mathbf{F}} = \left( \mathbf{I}_2 + \mathbf{i} \hat{\nabla} u_z \right)^{-1}.
\]

(2)

Here \(\mathbf{I}_2 = ii + jj\) is the in-plane identity tensor, and the expression for the gradient of deformation from the reference to the intermediate configuration \(\hat{\mathbf{F}}\) follows from \(\mathbf{F} = \hat{\nabla} \hat{\mathbf{r}} = \hat{\mathbf{F}}^T \cdot \hat{\nabla}(\hat{\mathbf{r}} - u_z \hat{z})\). The strain measures of a classical shell

\[
\mathbf{E} = \frac{1}{2} \left( \mathbf{F}^T - \mathbf{I}_2 \right), \quad \mathbf{K} = \mathbf{F}^T \cdot \mathbf{b} \cdot \mathbf{F}
\]

(3)

feature the actual second metric tensor \(\mathbf{b} = -\nabla \mathbf{n}\), and after mathematical transformations we express the tensor of bending strains with the operator of the intermediate configuration:

\[
\mathbf{K} = \mathbf{F}^T \cdot \mathbf{K} \cdot \mathbf{F}, \quad \hat{\mathbf{K}} = \hat{\nabla} \nabla \mathbf{r} \cdot \mathbf{n}.
\]

(4)

Now, the strain energy per unit area in the reference configuration is computed as a quadratic form

\[
U = \frac{1}{2} \left( A_1 (\text{tr} \mathbf{E})^2 + A_2 \mathbf{E} : \mathbf{E} + D_1 (\text{tr} \mathbf{K})^2 + D_2 \mathbf{K} : \mathbf{K} \right)
\]

(5)

with known coefficients, Ref. [3, 7]. The total strain energy

\[
U_{\Sigma} = \int_0^L \int_{-w/2}^{w/2} U(\det \hat{\mathbf{F}})^{-1} d\hat{y} \, dx
\]

(6)

is integrated in the intermediate configuration using the finite element discretization of displacements \(u_x, u_y, u_z\) and minimized.

3. Numerical benchmark problem

Prior to modeling the axial motion, we test the formulation by seeking the equilibrium of a trapezoidal plate of the width \(w\) and side lengths \(L\) and \(L + u_{10}\), see Fig. 3. The inclined edge is rotated parallel to the right one by kinematically prescribed displacements \(u_x, u_y\) such that the actual configuration is bounded by the lines \(x = 0\) and \(x = L\). The mapping Eq. (1) is thus possible with the intermediate configuration \(0 \leq \hat{y} \leq w\), which is discretized using \(C^0\) continuous finite element approximation of displacements, presented in Refs. [7, 8].

The compressed shell buckles out of plane, and the region with \(u_y < 0\) is "shadowed" by the gray initial configuration in Fig. 3. The transverse edges of the finite element mesh remain parallel in the deformed configuration. This corresponds to Eq. (1), as the mapping \(\mathbf{r}(\hat{F})\) includes only \(u_y\) and \(u_z\).

The considered parameters of the model in SI system are \(L = 1, w = 0.4\), thickness of the plate \(5 \times 10^{-3}\), Young’s modulus \(2.1 \times 10^{11}\) and Poisson’s ratio 0.3. In Table 1 we summarized the maximum and minimum values of the out-of-plane displacements, computed for various discretizations using the present method and the conventional shell finite elements with Lagrangian description, discussed in the above references. The current implementation of the mixed Eulerian-Lagrangian finite element formulation using Wolfram Mathematica is yet restricted concerning the size of the mesh, but one can conclude that the results converge to the same solution.

![Deformation of a trapezoidal plate](image)

**Figure 3:** Deformation of a trapezoidal plate, seen from above (together with the undeformed configuration) and from the side

<table>
<thead>
<tr>
<th>Discretization, (h_x \times h_y)</th>
<th>Mixed E.-L.</th>
<th>Lagrangian</th>
</tr>
</thead>
<tbody>
<tr>
<td>(4 \times 2)</td>
<td>-0.07270 0.18577</td>
<td>-0.07322 0.18138</td>
</tr>
<tr>
<td>(8 \times 4)</td>
<td>-0.05846 0.18427</td>
<td>-0.05831 0.18256</td>
</tr>
<tr>
<td>(16 \times 8)</td>
<td>-0.05542 0.18319</td>
<td>-0.05527 0.18259</td>
</tr>
<tr>
<td>(32 \times 16)</td>
<td>—</td>
<td>-0.05490 0.18262</td>
</tr>
</tbody>
</table>

**Table 1:** Mesh convergence and comparison of the mixed Eulerian-Lagrangian and traditional Lagrangian frameworks

References


MS03

Computational Mechanics of Concrete and Geomaterials

organized by J. Pamin, J. Tejchman and A. Winnicki
Experimental and numerical analysis of precast-monolithic building floors under in-plane loading

Piotr Alawdin¹, Alexander I. Mordich², Jury A. Muzychkin³

¹ Faculty of Civil Engineering, Architecture and Environmental Engineering, University of Zielona Góra
prof. Z. Szafrana 1, 65-516 Zielona Góra, Poland
e-mail: P.Aliawdin@ib.uz.zgora.pl

² OOO “BESTEngineering”
Kujbyshova 36, 220123 Minsk, Belarus
e-mail: Alex.Mordich@mail.ru

³ RUP "Institute BelNIIS", Minsk, Belarus
F. Skoriny 15 B, 220114 Minsk, Belarus
e-mail: wrdyanf@mail.ru

Abstract

The paper presents an investigation of a structural element of a floor slab of a residential building developed in Belarus and also widely used in construction in Russia and Ukraine. The structural element consists of reinforced-concrete flooring, embedded at end faces in a girt strip. This strip acts as a crossbeam for the slab floor. A full-scale experiment for a real structural element under diagonal load in its plane has been performed. Numerical analysis was carried out with the use of the software ANSYS based on the finite element method. A simplified analytical approach was also suggested for a stress-strain state structural analysis. The analytical and numerical results were compared with the experimental data. Identification of an analytical structural element model is proposed.

Keywords: precast-monolithic building floors, reinforced concrete structures, experiment, finite elements, analytical solution, identification

1. Introduction

Long-term experience in creating public and residential buildings in Belarus, Russia and Ukraine demonstrates that the precast hollow core slabs are effective to use as a part of mixed cast-in-situ and precast floor systems. In these systems the slabs are grouped in cells formed by crossed monolithic girders. The precast hollow core slabs in the reviewed floor system are propped upon concrete joggles created on side edges of bearing girders and set in the hollowness of slabs on their ends. Such leaning of slabs allows producing smooth ceilings and implementing open plan solutions in buildings without additional costs. Strength and deformability of such constructions under static and dynamic vertical (normal to the floor slab plane) loads were investigated earlier [1,2].

For such construction of floor slab the practical interest is in the behaviour of its in-plane shear. To answer this question, experimental investigations were carried out. The investigations involved testing up to failure of two natural structural elements of floor slab as well as natural tests to check the influence of vertical and horizontal loading of an 18-storey building frame in Belgorod (Russia) during the construction process [3]. Similar in-plane loading is used for the elements of both masonry and metal structures [4].

In the paper the main attention is paid to the numerical investigation of the floor slab construction structural element based on the finite element method, an approximate analytical solution and comparison of theoretical and experimental data obtained. The problem of identifying model parameters for this structure is analysed.

2. Full-scale structural element, the test procedure

A scheme of the structural element, its cross sections and load are shown in Fig. 1. Square-shaped in plan structural element consists of two precast hollow core slabs, with the inter-slab joint in the middle, as well as a monolithic concrete frame placed on the outer contour. Slabs with nominal width of 1.20 m and a thickness of 220 mm are simply supported at the ends on the concrete dowel formed integral with the lateral branches of the frame and placed in the cavities of the slabs to a depth of 100 ± 10 mm. Both the outer lateral side of the slab tightly adjoined to the longitudinal branches of the frame, and the 100 mm wide inter-slab joint between them were filled with concrete without reinforcing.

Before the loading started, deflection gauges 6PAO were fixed on every sample to measure average deformations of the compressed and stretched diagonal. Dial gauges with a measuring sensitivity of 0.01 and 0.001 mm were fixed along the seams; the gauges were used to measure the mutual displacement along the slab contact and the banding frame under the load. On every stage of the loading the specified parameters were probed, and the appearance of the cracks and damage in the concrete were registered. The relation between tensile and compressive strains of the structural element diagonal from the load is shown in Fig. 2 (lines 1 and 2).

Shear forces that appeared along every contact seam between monolithic girders and sides of hollow core slabs are received by crossing monolithic girders that cross them. For this reason, in cast-in-place and precast floor slabs of such a construction, with leaning of hollow core slabs on concrete joggles of bearing girders, mutual shear displacements of hollow core slabs and monolithic girders are not possible. In consequence, shear conditions on concrete carrying joggles in
the floor slab plane along contact seams cannot be obtained. As a result, the joint concreting floor slab formed by hollow core slabs and monolithic girders works as a continuous monolithic disk while affected by the horizontal load.

Figure 1: Geometry, cross sections 1-1, 2-2 and load of the structure; 1 - precast slab; 2 - inter-slab joint; 3 - monolithic concrete frame; 4 - dowel

3. Numerical simulations

Stress-strain state of the entire structural element with the floor slab embedded at end faces in a girt strip was also investigated on the basis of the finite elements method program complex ANSYS. The finite elements of the SOLID45 type (3-D STRUCTURAL SOLID) were used here. The total number of elements is 638575, and joints 130324, size edges of the element 2.8 cm. Only linear behaviour of the system was examined here, for the loaded diagonal it means \( V = F/\sqrt{2} < 1000 \) kN; for the unloaded diagonal \( V < 887 \) kN (whereas the maximal load carrying capacity of real structural element is a little more and equal \( V = 1081 \) kN, or \( F = \sqrt{2V} = 1529 \) kN). The results of numerical simulations for tensile and compressive strains of structural element diagonal versus load are shown in Fig. 2 (lines 3 and 4).

4. Analytical solution, analysis of results and model identification

Analytical solution is obtained for a generalized plane stress state of a structural element as a continuous and homogeneous elastic body. This solution uses the formulae of the theory of elasticity for the force acting on the diagonal of a truncated, infinitely long wedge.

Figure 2: Tensile and compressive strains vs load \( V = F/\sqrt{2} \): 1,2 - experimental and 3,4 - numerical data

The performed data comparison for the load, deformation and displacements, obtained with different methods, demonstrated that the analytical approach provides the lower bound of displacements in relation to the experimental and numerical approaches.

The procedure of identification of material parameters for the structure is suggested and used for the simplified model updating.

5. Conclusions

In mixed cast-in-place and precast floor slabs of such a construction and loading mutual displacements of hollow core slabs and monolithic girders are not possible; shear conditions on concrete carrying joggles in the floor slab plane along contact seams cannot be obtained. As a result, the joint concreting floor slab formed by hollow core slabs and monolithic girders works as a continuous monolithic disk under in-plane load.

References

Simulations of cracks in concrete with gradual transition from continuous to discontinuous description

Jerzy Bobiński¹, Jacek Tejchman²

¹,² Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: bobin@pg.gda.pl¹, tejchmk@pg.gda.pl²

Abstract

The paper presents a constitutive model for concrete which combines a continuous and discontinuous crack description to simulate the concrete under tensile dominated loadings. In a continuum regime, two different constitutive laws were used. First, a plasticity model with a Rankine failure criterion and an associated flow rule was used. Second, a constitutive law based on isotropic damage mechanics was formulated. Both alternatives were enriched by a characteristic length of micro-structure with the aid of a integral non-local theory to preserve mesh-insensitivity of FE-results. Displacement jumps across cracks were captured by applying eXtended Finite Element Method (XFEM) with cohesive tractions. A transition algorithm between a non-local continuum model and XFEM was formulated. A transfer function was introduced allowing for a gradual switch from a continuous (smeared) to discontinuous (discrete) softening process. Several benchmarks were numerically simulated with a dominated mode-I (e.g. uniaxial tension and bending) and under mixed-mode conditions.

Keywords: concrete, cracks, non-locality, XFEM

1. Introduction

Modelling of quasi-brittle materials like concrete is demanding task due to the presence of cracks. Cracks are responsible for the both strength and stiffness reduction and they precede the failure of the structure. At the beginning of loading, a region with several micro-cracks is formed. Later these micro-cracks create a macro-crack. An adequate description of cracks in numerical FE calculations is extremely important to obtain physically realistic results.

Within continuum mechanics, there exist two main approaches to describe cracks. The first one describes them in a smeared sense as localized zones of micro-cracks with a certain finite width. The material behaviour may be described using e.g. elasto-plastic, damage mechanics or smeared crack and coupled constitutive laws. These formulations include softening, so they have to be equipped with a characteristic length of microstructure to preserve the well-posedness of the boundary value problem. As an alternative, displacement jumps (discontinuities) along cracks may be introduced while keeping the remaining region as a continuous one. The oldest solutions used interface elements which were defined along finite element edges. The modern ones allow for considering displacement jumps in the interior of finite elements using the embedded discontinuities or XFEM. A smeared approach is more appropriate when describing a micro-crack formation process while a discontinuous one allows for a more realistic simulation of discrete macro-crack propagations. Usually, only one approach is used to simulate a fracture process in concrete during the entire deformation process. A combination of continuous and discontinuous approaches make it possible as in experiments to realistically capture all stages of fracture.

2. Numerical model

2.1. Continuum

In continuum, an elasto-plastic constitutive law with standard Rankine yield criterion was used first. An associated flow rule was assumed. To define softening under tension a linear or an exponential curve was used. The second constitutive law was defined within continuum isotropic damage mechanics. The initiation and propagation of micro-cracks was described by a scalar damage parameter $D$. Several different definitions of the equivalent strain measure was examined. An exponential evolution of the damage variable $D$ was adopted.

To obtain mesh-independent results, a non-local theory in an integral format was used for both laws as a regularization technique. A Gaussian function was chosen to calculate the weighing function with a characteristic length of the microstructure. This length was reduced near boundaries using a distance-based scaling method ($|1|$).

2.2. Discontinuity

To describe displacement jumps in continuum, XFEM was chosen based on the Partition of Unity (PUM). In the continuum body, a linear elastic constitutive law was assumed. Tractions along crack were calculated using assumed discrete cohesive laws. In the normal direction, a linear or exponential softening was assumed. During unloading, the secant stiffness was used with a return to the origin. In a compressive regime, the penalty stiffness in the normal direction was used. In a tangential direction, a linear relationship between a displacement jump and traction was defined.

2.3. Transition

Initially, in all finite elements (integration points), a continuum constitutive law with non-local softening was solely active. A new crack segment was created in the element if the
value of the softening driving (state) variable of the constitutive model used was greater than assumed threshold value in any of element's integration points at the front of a crack tip. The direction of the crack segment in the finite element was calculated from the field of the state variable at the front of the crack tip. The tractions at the moment of cracking and softening curve parameters were determined from two principles: the dissipated energy balance and stress equilibrium.

To avoid convergence problems observed in the previous version of this coupled model [2, 3], the following improvement was proposed. Along the discrete crack, a band with doubled finite elements and nodes was introduced (Fig. 1). The width of this band was chosen in such a way to include the width of the continuum localisation zone. In the “bottom layer” of finite elements, a continuum law with softening was applied. In the “top layer” of finite elements, linear elasticity in bulk and discrete crack with cohesive softening law was used. Both doubled element sets share the same nodes along zone boundaries. The resultant stresses are calculated as:

\[ \sigma = (1-\rho)\sigma_c + \rho \sigma_D \]

where \( \sigma_c \) are the stresses in the bottom (continuous) layer, \( \sigma_D \) are the stresses in the top (discontinuous) layer and \( \rho \) is the transfer function. This function starts from 0 (no XFEM influence) and it grows till the value of 1 (only XFEM solution). Its growth depends on the opening of the crack segment.

To verify the proposed formulation, several benchmarks were numerically simulated with a dominated mode-I (e.g. uniaxial tension) and under mixed-mode conditions. Among them a three-point bending test was also studied. Figure 2 presents the geometry of the beam and the boundary conditions. The elasto-plastic model with exponential softening was used in continuous description of cracks. At the XFEM side, an exponential softening was assumed also. The transition point was defined at the level of 50 percent of the tensile strength. The calculated force – displacement curve is shown in Fig. 3. A very smooth response with no bumps and sudden drops was obtained that confirmed the correctness of our algorithm.

4. Final remarks

The presented constitutive law allows for a more realistic description of cracks in concrete elements. The formulation is quite general and it allows for using any constitutive laws in bulk continuum and any displacement jump – traction relationships along a crack. It may be also easily extended to take into account a decrease of a characteristic length upon loading and to obtain more realistic displacement profiles in a localized zone.

References


Distribution of damage in unconventionally reinforced concrete slabs subjected to impact loads

Krzysztof Cichocki¹, Mariusz Ruchwa*¹²

¹²Faculty of Civil Engineering, Environmental and Geodetic Sciences, Koszalin University of Technology
Sniadeckich 2, 75-453 Koszalin, Poland
e-mail: krzysztof.cichocki@tu.koszalin.pl¹, mariusz.ruchwa@tu.koszalin.pl²

Abstract

The study concerns numerical analysis performed in order to evaluate the final distribution of damage developed in circular plates made of fibre reinforced concrete with brick rubble aggregate subjected to impact loads. Adequate experimental tests have been carried out on the entire set of slabs produced with various types and amount of fibres. The development of damage, its distribution and intensity was observed and documented for each slab defining the mechanism of this phenomenon. Numerical discrete models were built and applied to simulate the entire problem using an advanced finite element method approach.

Keywords: experimental tests, fibre reinforced concrete, waste aggregate, impact loads, finite element method

1. Introduction

One of the goals of the study is to investigate the possibility to reuse the waste material (sorted brick rubble) as aggregate in the production of concrete elements [1]. Application of steel fibres (unconventional reinforcement) increased considerably the resistance and durability of such elements for impact loads. This is a very important characteristic for structures exposed to this load: elements of bridges, roadways, protective structures applied in order to minimize the effects of explosions and impacts, etc. The necessity to improve the impact resistance for concrete structures used in many areas of civil engineering was the main reason to undertake this research. Various factors and phenomena have been studied during experimental tests performed on series of circular plates, both qualitative and quantitative, as the failure mode of various types of plates, the number and dimensions of cracks propagated through the material, development of damage, etc.

Experimental tests carried out during the research served as a basis to create a discrete numerical model of the Finite Element Method in ABAQUS computer code environment [2]. Thus it was made possible to calibrate the parameters of applied numerical material model for concrete, describe the nature and characteristics of impact loads, define the applied algorithms of contact between the impacting objects and concrete element under investigation, as well as other parameters of numerical analysis in order to obtain an adequate and reliable numerical discrete model [3-6].

2. Experimental tests

A special experimental stand was built in order to perform the impact tests. All tests were carried out on circular slabs of 1 m diameter, thickness 0.1 m, with a various percentage of steel fibres. A detailed description of the experiment is provided in [1], the results are also discussed. An example of final distribution of damage for the slab reinforced with 0.5% of fibres volume is given in Fig. 1. This pattern of cracks traversing the entire thickness of the slab was obtained after 7 impacts of 40 kg weight (free fall from a 1 m height). For other reinforcement configurations the results were different, especially for the highest percentage of reinforcement (1.5%) and good quality fibres the pattern presented numerous cracks developed around the contact zone with the impacting weight.

Figure 1: Bottom view of a damaged plate

3. Numerical analysis

Adequate numerical models for the tested plates were created, based on a Finite Element Method (FEM) approach. Due to an impulsive character of the applied load, the explicit procedure of integration for equations of motion was used, with the application of ABAQUS/Explicit [2]. The main assumptions concerning all variants of the slabs were identical, the only difference concerned the description of material characteristics for an analysed group of slabs. For each model identical slab dimensions were considered constant (diameter 1 m, thickness 0.10 m), the same type of supports (modelled as rigid,
nondeformable surfaces) and the same type of impacting load, in order to obtain the most realistic scenario for the tests. Concrete slabs and impactor were modeled as three-dimensional deformable solid bodies, the supports were discretized with quadrilateral rigid elements. The master-slave type of contact between impactor and slab, as well as between slab and supports was introduced. A schematic view of the numerical model is shown in Fig. 2.

Figure 2: Schematic view of the numerical model

Figure 3: Distribution of damages for reinforced plate after first impact – bottom view

The final response of slabs depends on the characteristics of constituent materials and the quantity and distribution of fibres in the volume. The adequate proprietary code (script) to create the numerical model was prepared by the authors using Python programming language [2], included in ABAQUS computer code environment. As a result, the script creates an entire discrete model of the problem (input file for FEM computer code ABAQUS/Explicit), with all geometric, material and other necessary data, where the distribution of fibres was randomly generated by a pseudo-random number generator. The random position of each fibre was defined by coordinates of its centroid and adequate angles of inclination with respect to assumed reference system axes. Additionally the information about spatial distribution of fibres in such kinds of structural elements was taken into account (e.g., contribution of fibres with various inclination angles, distribution in thickness, etc.). All fibres were modelled using three-dimensional truss finite elements. Adequate interaction between concrete matrix and steel fibres was made possible by means of insertion of randomly distributed “cloud of finite elements” of fibres into concrete volume of solid elements and application of embedded elements definition, which allows for the interaction of two different meshes of finite elements, one inserted into another.

An example of numerical results is given in Fig. 3, with well visible zones of a totally damaged material, very close to distributions obtained in experimental tests.

4. Summary

Two main goals were the object of this study:
- experimental investigation on dynamic response of slabs subjected to repetitive impact loading up to total damage of material;
- numerical simulation of experimental tests, comparison of results, etc.

The experiments allowed to describe the development of damage patterns in a function of number of impacts. Additionally the secondary effects revealed during the entire phenomena were registered and documented.

Numerical simulations performed using the advanced features of FEM code ABAQUS show the adequate correspondence between experimental and numerical results, very useful for future practical applications.

The experimental and numerical results obtained in the study allowed for a better understanding of the specific behaviour of investigated structural elements subjected to impact loads. A detailed description of the final conclusions and remarks is given in [1] together with recommendations concerning the practical use of such elements.

References

Mathematical model of concrete degradation due to the alkali-silica reaction at the mesoscopic level

Witold Grymin¹, Marcin Koniorczyk², Dariusz Gawin³

¹,²,³ Department of Building Physics and Building Materials, Technical University of Lodz
Al. Politechniki 6, 90-924 Łódź, Poland
e-mail: witold.grymin@p.lodz.pl¹, marcin.koniorczyk@p.lodz.pl², dariusz.gawin@p.lodz.pl³

Abstract

A mathematical model of the alkali-silica reaction at the mesoscopic level was presented. In the model five governing equations (of dry air mass, water mass, energy, linear momentum and alkali mass) were defined. The model takes into account diffusion of alkalis from the cement paste towards the aggregate and binding of the alkalis by the gel. The diffusion of the alkalis depends on temperature, pore saturation and internal structure of the pores. The alkali-silica reaction takes place only in the aggregates, after the alkali concentration threshold was reached.

Keywords: alkali-silica reaction, alkali diffusion, chemical damage of concrete

1. Introduction and motivation

The alkali-silica reaction (ASR) is a chemical reaction, which takes place between the non-crystallized silica, to be found in particular types of aggregates, and alkalis, which are usually derived from the cement paste. The alkali-silica reaction is influenced by a number of factors: environmental conditions (i.a. temperature, humidity, alkali sources), type of aggregate, cement and additives used, concentration of alkalis, etc. The mathematical model and analysis presented in the paper contribute to a better understanding of the mechanisms of the reaction at the level of aggregate.

2. Description of the alkali-silica reaction at the mesoscale

The majority of models concerning the alkali-silica reaction is based on a phenomenological approach, the parameters describing rate of the ASR development are determined based on the experimental results. In this model, the rate of the reaction is influenced by the diffusion of alkalis into the aggregate. The ions are transported from the cement paste towards the aggregates. The alkali ions (Na⁺ and K⁺) are of particular importance to this reaction. The reaction takes place only if a certain threshold alkali concentration, corresponding to a certain pH value, has been reached. The value of a threshold alkali concentration depends on the activity of the aggregate.

Majority of the alkalis are derived from the cement paste during the hydration process, and usually analyses of the alkali-silica reaction in concrete are limited to this alkali source. However, the alkalis can be also released from certain aggregates and supplementary cementing materials or delivered from the environment. During the analysis of a alkali-silica reaction it has to be emphasized, that in the laboratory conditions, where the specimens are usually stored in water or over its surface, the alkalis can be leached from the specimens.

3. Mathematical model and results

The presented mathematical model is based on the model of the alkali-silica reaction in variable hygro-thermal conditions [3], in which four governing equations (water mass conservation, dry air mass conservation, energy conservation and linear momentum conservation) were defined. The solid skeleton voids are filled partly by liquid water and partly by a gas phase, which is an ideal mixture of dry air and water vapour. The governing equations were defined using the Volume Averaging Theory proposed by Hassanizadeh and Gray [4,5]. The chosen primary variable are: gas pressure $p^g$, capillary pressure $p^c$, temperature $T$, alkali content $c_a$, and displacement vector $u$. ASR reaction extent, $\Gamma_{ASR}$, is an additional internal variable of the model.

In the model additional mass balance equation, alkali ions conservation, is written in the following form:

$$-n\beta_s \frac{\partial T}{\partial t} + \frac{n}{\partial t} \frac{D_S}{\partial t} + \frac{n}{c_a} \frac{D_c}{\partial t} + \frac{1}{c_a \rho_s} \partial \cdot \mathbf{J}_{\text{alkali}} +$$

$$+ \frac{1}{c_a \rho_s} \partial \cdot (c_a \rho_s \mathbf{w}_{\text{ASR}}) - \beta_s (a - n) \frac{\partial p^g}{\partial t} +$$

$$+ \alpha \partial \mathbf{v} + \frac{1 - n}{\rho_s} \left( \frac{\partial \rho^c}{\partial t} \frac{D_m_{\text{gel}}}{\partial t} + \frac{\partial \rho^c}{\partial \varepsilon_{\text{ASR}}} \frac{D e_{\text{ASR}}}{\partial t} \right) =$$

$$= \dot{m}_{\text{water}} + \dot{m}_{\text{water-gel}} - \dot{m}_{\text{water-out}} - \dot{m}_{\text{alkali}}$$

where $s$, $w$, $a$ denote solid skeleton, water and alkalis, respectively, $n$ is the porosity, $\beta_s$ and $\beta_c$ are thermal dilatation coefficients of water and solid, $T$ is temperature, $S_c$ is saturation of pores with water, $c_a$ is concentration of alkalis, $\rho$ is the density, $\mathbf{J}_{\text{alkali}}$ is the diffusive flux of alkalis in the pore solution, $\mathbf{v}$ and $\mathbf{w}_{\text{ASR}}$ is velocity of the skeleton and of water with respect to the skeleton, $\alpha$ is the Biot coefficient, $m_{\text{water}}$ is mass of the alkali-silica gel, $\varepsilon_{\text{ASR}}$ is expansion caused by the ASR, and $\dot{m}_{\text{water}}$, $\dot{m}_{\text{water-gel}}$ and $\dot{m}_{\text{water-out}}$ are the masses of alkalis and water bonded by the gel, accordingly. The alkali-silica gel is considered as a part of the skeleton.
The diffusive flux of alkali ions in the pore solution is described by the Fick law:

$$J_{\text{alkali}} = -\rho D_{ij} \nabla c_i,$$

(2)

where $D_{ij}$ is the alkali ions diffusion tensor, defined as [6,7,9]:

$$D_{ij} = -n \frac{\delta}{\tau^2} \cdot e^{\alpha(t - t_0)} \cdot S_{ij},$$

(3)

where $D_{ij}$ is alkali diffusion coefficient in water ($D_{ij}^{\text{alkali}}$), $\delta$ is constrictivity, $\tau$ is tortuosity, and $\alpha$ and $\lambda$ are material parameters equal to 0.028 and 4.5, accordingly.

The ASR takes place in the aggregate after reaching a threshold value of the alkali concentration, $c_{\text{alkali,lim}}$. The ASR extent rate is written in a rate form as:

$$\dot{\gamma}_{\text{ASR}} = \begin{cases} 1 - \Gamma_{\text{ASR}} c_i, & c_i \geq c_{\text{alkali,lim}}, \\ 0, & c_i < c_{\text{alkali,lim}} \end{cases},$$

(4)

where $t$ is the characteristic time of the reaction.

The strains are described in the model as functions of the effective stresses. The material damage due to the alkali-silica reaction is modelled with the Mazars’ non-local formulation of isotropic damage theory. The model equations are discretized in space using finite element method and in time using finite difference method. The following equation set is obtained:

$$C_{ij}(\Delta \mathbf{x}^{n+1} - \mathbf{x}^n) + K_{ij} \mathbf{X}^{n+1} - \mathbf{f}_i = \mathbf{f}_j,$$

(5)

where

$$K_{ij} = \begin{bmatrix} K_{gg} & K_{gc} & K_{gg} & K_{gg} \\ K_{cg} & K_{cc} & K_{cg} & K_{gg} \\ K_{gc} & K_{cg} & K_{gg} & K_{gg} \\ K_{gc} & K_{cg} & K_{gg} & K_{gg} \end{bmatrix}, \quad \mathbf{f}_i = \begin{bmatrix} f_{i1} \\ f_{i2} \\ f_{i3} \\ f_{i4} \end{bmatrix}.$$

(6)

The non-linear equation set is solved using the Newton-Raphson method and monolithic approach. The numerical model has been applied in order to analyse development of the alkali-silica reaction in aggregate of diameter equal to 2, 4 or 8 mm embedded in the cement paste. The alkalis, present initially mainly in the cement paste, diffuse towards the middle of the aggregate (Fig. 1B). The reaction is initiated (Fig. 1A) when the alkali threshold has been reached (Fig. 1B). Simultaneously, in the places where the alkalis were bonded by the arising gel (see Fig. 1B, D = 8 mm), the reaction is stopped (Fig. 1A).

**References**


Abstract

In the paper comparative analysis of RC beams strengthened by CFRP strips using the Finite Element Method (FEM) and the eXtended Finite Element Method (XFEM) is presented. A good agreement of results obtained by means of FEM with the laboratory experiments was observed and reported in previous papers [4,5]. The application of FEM analyses have proved to be effective in the assessment of the load carrying capacity of RC beams strengthened by CFRP strips. The application of Concrete Damage Plasticity (CDP) model of RC beam allowed to predict a layout and propagation of cracks leading to failure (intermediate crack debonding failure). The XFEM allows to model propagation cracks through the elements under mixed-mode loading conditions therefore it can be an effective tool to simulate the complex behaviour of CFRP strengthened concrete elements. The preliminary study was carried out and first results were presented.

Keywords: CFRP strips, composite materials, FEM and XFEM analysis, load carrying capacity of RC beams

1. Introduction

Strengthening of reinforced concrete elements by bonding composite strips or mats of carbon fibers (CFRP) to their tension and/or shear zones is nowadays a well-known alternative to traditional techniques. One of the failure modes of CFRP-strengthened concrete beams is the intermediate crack debonding of CFRP initiated at the tip of flexural or flexural/shear cracks in concrete substrate, location at which the interface is subjected to mixed-mode loading. This mode of failure was obtained in series of experiments performed on RC simply supported beams strengthened using CFRP strips in their flexural zones at different levels of preloading [4,5]. Numerical analyses of the considered RC beams were performed using the Finite Element Method [4,6].

The presence of cracks in concrete elements causes that conventional methods of modelling, including FEM, turn out to be inadequate and not enough efficient to represent some aspects of real behavior of beams before failure. In the paper the idea of numerical analysis method using the eXtended Finite Element Method is presented to model CFRP intermediate crack debonding failure started at the tip of a flexural crack in RC beam.

2. The FEM analysis

In the previous author’s FEM analyses all numerical models of beams have been created and carried out in the environment of the Abaqus/Standard code. The Concrete Damage Plasticity model (CDP) was used to describe behavior of concrete [7]. It is a continuum, plasticity-based, damage model for concrete. It can be used for plain concrete, even though it is intended primarily for analysis of RC structures.

In the analyses it was assumed that the reinforced concrete can work in tensile zones even after cracks occurrence (‘tension stiffening’ effect). In order to represent the behavior of concrete after cracking the description of concrete was used expressed by a function of the fracture energy $G_f$ released in the process of formation of cracks. The strain-softening behavior of concrete in tension was expressed by the stress-displacement relationship ($\sigma$-$w$) according to the concept of fictitious cracks model of Hillerborg [3]. Thus, tension stiffening effect was presented applying the fracture energy cracking criterion. Additionally, this approach made it possible to account the effects of reinforcement interaction with concrete to be simulated in a simple manner. The reinforcement bars were modeled in a discrete manner as truss elements embedded in 2D or solid 3D elements of concrete.

Numerical model validation and verification procedure of numerical modeling based on the results of laboratory tests have been completed successfully [4,5]. The FEM proved to be useful in the determination of load carrying capacity levels as well as the stiffness of strengthened element at every level of loading. The identification of crack arrangement for all tested beams was made on the basis of maps of damage parameter $d$ which only mimics cracks. The modes of failure of beams (including propagation of delamination of strips) were also difficult to determine. The applied approach allowed, after introduction of failure criteria, to follow the whole RC beam behavior up to collapse. However, the intermediate crack debonding failure observed during the experiments was not possible to simulate using the standard FEA modeling. The Extended Finite Element Method is employed to simulate intermediate FRP debonding along the FRP–concrete interface.

3. The XFEM analysis

The XFEM is an alternative method to the FEM which extends the allowable basis functions known as partition of unity methods. When applied to crack propagation problems, the XFEM introduces two extra sets of functions in addition to standard FEM nodal basis functions: a step function $H(x)$ to capture the discontinuity in displacement across a crack and a set of functions $F(x)$, typically expressed in polar coordinates centered on the crack tip, that capture stress singularity in that region. The extra basis functions are defined as the product of these enrichment functions with the standard nodal basis
function in order to ensure that the basis of functions remain mesh-based and local to the enriched nodes.

The XFEM was proposed first in context of fracture by Belytschko and Black [1] and is nowadays successfully used for analysis of phenomena of concrete cracking elements as well as debonding of FRP from the surface of concrete [2,8].

This method does not require the mesh to match the geometry of discontinuities. It can be used to simulate initiation and propagation of a discrete crack by using fracture energy criterion along an arbitrary, solution-dependent path in the bulk material without the requirement of remeshing (crack propagation is not tied to the element boundaries in a mesh).

The XFEM allows the presence of discontinuities (cracks) in the elements of a finite element analysis by enriching the element with additional degrees of freedom. This technique can model crack opening and increase accuracy of approximation near the crack tip. The degree of freedom enrichment is done with special displacement function consisting of usual nodal shape function as well as the associated discontinuous jump function across the crack (Heaviside distribution) and the asymptotic crack tip functions.

The failure mechanism including degradation and eventual separation between two surfaces consists of two components: a damage initiation criterion and a damage evolution law. The damage initiates when the contact stresses and/or contact separations satisfy the damage initiation criterion. The damage evolution law describes the rate of cohesive stiffness degradation after the initiation criterion has been reached. In analysis concerning phenomena of debonding of FRP strips from the RC concrete surface the maximum principal stress was assumed a damage initiation criterion and the fracture criterion described by a value of fracture energy $G_f$ as a damage evolution law.

4. Preliminary numerical results

Before applying the XFEM to analyse the intermediate crack debonding of FRP strengthened RC beams the validation of the model was performed based on the experimental results obtained in [4] for concrete beams only. The four-point-bending case of concrete beams was studied first (Fig. 1).

![Figure 1: Concrete beam used for XFEM analysis [cm]](image1)

In the series of experiments the mean ultimate force was measured as 31.0 ±2.45kN whereas in the XFEM it was read as 38.4 kN. The mode of failure was determined in both studies by a single crack occuring close to the middle of beam (Fig. 2). The concrete beam model was as 2D plane strain CPE4R elements. The elastic material model was used to describe concrete behavior with the criterion of crack initiation as the maximum principal stress. In numerical simulations it was not necessary to predefine in layout of mesh beginning of crack for further propagation.

The application of the XFEM to intermediate crack debonding failure will be presented during the congress.

5. Conclusions

The application of the XFEM for analysis of brittle materials like concrete is very effective while taking into account local issues. Therefore this method may also be used to study the mechanisms of intermediate crack debonding failure. In such a case it is a good alternative for standard FEM formulation based on continuum mechanics.

References


Modelling the strains induced by Delayed Ettringite Formation in cement-based materials

Marta Januszkiewicz*, Francesco Pesavento*, Witold Grymin*, Dariusz Gawin**

1,3,4 Faculty of Civil Engineering, Architecture and Environmental Engineering, Lodz University of Technology
Al. Politechniki 6, 90-924, Lodz, Poland
e-mail: marta.j.90@gmail.com 1, witold.grymin@p.lodz.pl 3, dariusz.gawin@p.lodz.pl 4

2 Department of Civil, Environmental and Architectural Engineering, University of Padovavia Marzolo 9, 35131 Padova, Italy
e-mail: francesco.pesavento@dicea.unipd.it

Abstract

The paper deals with Delayed Ettringite Formation (DEF), which is a type of chemical degradation of moist cement based materials due to expanding phases produced by the reaction. A phenomenological, rate-type model of DEF strains, considering effect of the temperature and duration of heat-treatment of the material at early ages, and the ambient temperature and relative humidity history, on the development of strains is introduced into the mathematical model of heat and moisture transport in deformable porous media. The model equations are solved numerically by means of finite element, finite difference and Newton-Raphson methods. The computer code was developed and used for the model validation by comparing its results with some published experimental data.

Keywords: DEF-induced strains, chemical deterioration, cement based materials, kinetics of DEF, hygro-thermal conditions

1. Introduction

Delayed Ettringite Formation is the reaction resulting in formation of ettringite within a cementitious material that begins several months (or even years) after its setting and without any contribution of any external sulphate [1]. Most commonly it arises in concrete cured at elevated temperatures at early ages or massive concrete elements, temperature of which exceeded the threshold of 65-70°C due to the self-heating effect of the temperature and saturation degree, is assumed to be proportional to the reaction rate,

\[ \varepsilon_{\text{DEF}}(t) = \beta_{\text{DEF}} \cdot (S_e, T_0, t_0) \cdot \Gamma_{\text{DEF}}(t) \cdot I, \]  

where \( \beta_{\text{DEF}} \) is the material parameter, dependent on the actual water saturation degree, \( S_e \), the temperature, \( T_0 \), and duration, \( t_0 \), of heating treatment of concrete at early ages.

The reaction rate, \( \Gamma_{\text{DEF}} \), dependent on the actual temperature and saturation degree, is given by,

\[ \Gamma_{\text{DEF}} = \frac{1 - \Gamma_{\text{DEF}}}{t_r}, \]  

with the characteristic time of reaction, \( t_r \), described by the following relationship, similar to that for the ASR [4],

\[ t_r = \tau_r(T) \cdot \frac{1 + \exp\left(-\frac{\tau_r}{\tau}\right)}{\Gamma_{\text{DEF}} + \exp\left[-\frac{\tau_r}{\tau}\right]}, \]  

where \( \tau_r(S_e, T) \) is the latency time, related to the time until the beginning of the swelling,

\[ \tau_r(S_e, T) = \tau_{r,0} \cdot \exp\left[U_r \cdot \left(\frac{1}{T} - \frac{1}{T_0}\right)\right] \cdot (A_r \cdot S_e + B_r), \]  

and \( \tau_r(T) \) is the reaction time, corresponding to the rate of development of the swelling [4],

\[ \tau_r(S_e, T) = \tau_{r,0} \cdot \exp\left[U_r \cdot \left(\frac{1}{T} - \frac{1}{T_0}\right)\right] \cdot (A_r \cdot S_e + B_r) \cdot \exp\left(-\frac{\tau_r}{\tau}\right). \]

where \( \tau_{r,0} \) are the reaction and latency time constants at the temperature \( T_0 \), \( U_r \) and \( U_r \) are the activation energies for the reaction and latency time, accordingly, \( A_r \), \( B_r \), \( A_r \) and \( B_r \) are the material parameters determined experimentally.

2. DEF induced strains

The evolution of DEF strains in time has similar shape as for the ASR strains, hence the proposed model has a similar form as the description of the latter ones [4]. The rate of DEF strains, \( \varepsilon_{\text{DEF}} \), is assumed to be proportional to the reaction rate,
The primary variables of the model are: gas pressure, $p_g(x,t)$, capillary pressure, $p_c(x,t)$, temperature, $T(x,t)$, displacement vector $u(x,t)$, and $\Gamma_{DEF}(x,t)$ is the internal variable.

The mathematical model consists of four governing equations: water mass conservation, dry air mass conservation, energy conservation and linear momentum conservation, as well as the evolution equation describing DEF reaction kinetics. The model equations are numerically solved by means of finite element and finite difference methods [5], by means of the research computer code developed by the Authors.

4. Model validation

The model validation was performed by comparison the simulation results with the DEF strains measured by Martin [2] for the concrete specimens exposed to different heat-treatment during maturing at early ages: 1) heated to temperature $T=80^\circ C$ for 3 days, and then stored in different humidity conditions (i.e. kept in water or above the water surface), Fig. 1; 2) for three different durations of heating treatment at 80$^\circ C$ (i.e. 1, 3 or 5 days), Fig. 2; 3) heated for 3 days to three different temperatures (70$^\circ C$, 80$^\circ C$ and 85$^\circ C$), Fig. 3.

Figure 1. The comparison the simulation results for 2 different moisture states of specimens with the experimental data [2]

Figure 2. The comparison of simulation results for 3 different durations of heat treatment with the experimental data [2]

Figure 3. The comparison of simulation results for 3 different temperatures of heat treatment with the experimental data [2]

Figure 4. The comparison of simulation results for 2 different temperature of DEF process with the experimental data [6]

As the second stage of the model validation, the effect of ambient temperature on the DEF strains progress was analysed. The strains measured for 20$^\circ C$ and 38$^\circ C$ [6] were compared with the simulations, Fig. 4.

It can be observed in Figs 1 – 4, that the DEF strains predicted by the proposed model are in a good agreement with experimental results, sufficient for practical applications.

References

Modelling the frost-induced damage in fully saturated cement-based materials

Marcin Koniorczyk1, Dariusz Gawin2*

1,2 Faculty of Civil Engineering, Architecture and Environmental Engineering, Lodz University of Technology
Al. Politechniki 6, 90-924, Lodz, Poland
e-mail: marcin.koniorczyk@p.lodz.pl1, dariusz.gawin@p.lodz.pl2

Abstract

The paper considers water freezing/thawing in cement-based materials. The influence of pressure, temperature and the size of pores on the equilibrium between solid and liquid water was investigated using Gibbs free energy. A mathematical model was proposed describing heat and water transport in deformable porous materials considering water phase change kinetics. The rate of water freezing is proportional to the affinity, which is given by means of the ice-liquid water interface curvature. The frost-induced damage was modelled using the delayed damage approach. The numerical code was developed using finite element, finite difference and Newton-Raphson methods.

Keywords: frost-induced damage, crystallization pressure, concrete, delayed damage, multiphase domain

1. Introduction

Frost damage due to cyclic water freezing/ice thawing in the pores of concrete, is one of the main reasons endangering durability of the structures in cold climates. For this reason, for several decades showed this topic a subject of scientific research. Already in 1945 Powers formulated his famous hypothesis, which was able to explain some issues related to frost resistance of concrete, in particular, the effect of material micro-structure on its frost damage. From that time several theories, mostly based on thermodynamics of liquid crystallization in porous media were formulated, e.g. by Everett [1], Bazant [2], Setzer [3]. In the present paper, a mathematical model of coupled heat and moisture transfer is formulated by means of mechanics of multiphase porous media, previously proposed by Gawin and Schrefler [4], extended here to consider water freezing/melting and crystallization pressure exerted by the process, and material frost deterioration possibly caused by the pressure. From the numerical viewpoint, modelling these water phase changes, associated by considerable heat sources/sinks, constitutes rather difficult problem, which can be solved by means of different techniques, e.g. using apparent thermal capacity, using very stable finite difference time schemes or formulating an appropriate Stefan’s problem. Here the phase transformations water – ice is modelled by means of kinetic, non-equilibrium approach [5].

2. Water phase change

Water phase change (freezing/melting) in a capillary-porous material is a very complex physico-chemical process which is rather difficult for mathematical modelling In modelling the assumption that exists there, thermodynamic equilibrium between the solid (ice) and liquid phases of water is usually applied. Here the water freezing / melting in a capillary-porous material is modelled as a non-equilibrium process (but close to thermodynamic equilibrium), with its kinetics is governed by a linear evolution law, obtained from the second law of thermodynamics in the following form [3],

\[ \dot{s}_i = A_{fr,eq} \eta_c \]  

where \( \dot{s}_i \) is the rate of internal production of molar entropy due to the freezing, \( \eta_c \) the rate of freezing, and the process affinity is given by the following equation [3],

\[ A_{fr,eq} = \mu_c(T, p^\ell) - \mu_l(T, p^s) \]  

with \( \mu_l \) and \( \mu_c \) being chemical potentials of liquid water and ice (crystallized water), respectively.

From linear thermodynamics of chemical reactions it is well known that the reaction rate is proportional to the chemical affinity, hence for freezing one obtains,

\[ \dot{\eta}_c = k A_{fr,eq} \]  

Since the freezing rate is dependent on the micro-flows of water and heat during reaching thermodynamic equilibrium between liquid water and ice, and the proportionality constant, \( k \), is given by means of the characteristic time of freezing, \( \tau_{fr} \), one obtains:

\[ \dot{\eta}_c = \frac{1}{RT \tau_{fr}} A_{fr,eq} = \frac{\nu_l}{RT} \frac{\Sigma_a (T_a - T)}{\tau_p} \]  

where \( p \) is pressure, \( T \) is temperature, \( \nu_l \) is molar volume of liquid phase and \( \Sigma_a \) denotes enthalpy of water solidification.

Crystallization pressure

During freezing of water in a saturated porous material, the pressure is exerted on the pore walls by the liquid water and ice crystals. It is assumed that the isotropic pressure in the crystal is controlled by the curvature of interface between liquid and crystal, which are in equilibrium. Considering the difference in the chemical potential of small and large crystals of cylindrical shape, Everett [1] derived the equation:

\[ p^s = p^l + \gamma_{cl} \frac{dA}{dV} = p^l + 2 \gamma_{cl} \left( 1 - \frac{1}{R_{eq}} \right) \]  

where \( R_{eq} \) is the equilibrium pore radius, associated with the overcooling temperature \( T \), is the pore side radius. The pores geometry is described by the cumulative pore size distribution curve \( P(r) \), with \( P(\infty) = 0, P(0) = \phi \), where \( \phi \) is the open porosity.

*Research has been financed by NSC Poland, grant no. UMO-2011/03/B/ST8/05962.
Assuming that the advancing ice front is hemispherical, the volume averaged pressure exerted by ice on solid walls reads:

$$\langle \eta_r p^s \rangle = \frac{R^2}{\phi} \int_0^\infty \left( \frac{2}{R} - \frac{1}{r} \right) V(r) \frac{dr}{r}$$

(6)

3. Mechanical equilibrium

The total stress of a saturated material, $t_{\text{total}}$, consists of the effective stress, $t'$, transferred by the solid skeleton and a part, which accounts for the pressure exerted by the pore fluids $p^S$:

$$t_{\text{total}}' = t' - b p^S$$

(7)

where $b$ is Biot’s number, $K_T$ is the bulk modulus of porous material and $K_S$ is the bulk modulus of solid skeleton. The pressure exerted by the phases occupying pores on the solid skeleton is equal to: $p^S = \eta_r p^s + \eta_c p^c$. In general, a total strain of a cement based material is decomposed in the elastic and thermal part, so the effective stress, considering damage is, [6]:

$$\varepsilon = (1 - d)D(\varepsilon_{\text{w}} - \varepsilon_c)$$

(8)

For transient problems the damage rate, $d$, is determined by the fundamental equation of the delayed damage model, which reads

$$\frac{d}{\tau_c} = \left[1 - \exp\left(-a[g(x) - d]\right)\right]$$

(9)

where $\tau_c$ is the characteristic time of damage, representing the inverse of the maximum damage rate, and $a$ is the second parameter of the model.

4. Frost induced damage of concrete wall

The example concerns the performance of a concrete wall, fully saturated with water, exposed to variation of external temperature causing cyclic material freezing / thawing. The hydro-thermal behaviour and the stress-strain field in the fully saturated concrete wall were numerically analysed.

![Figure 1: The change of volume averaged crystallization pressure in the time domain](image)

![Figure 2: The change of damage coefficient in the time domain](image)

Figure 1: The change of volume averaged crystallization pressure in the time domain

Figure 2: The change of damage coefficient in the time domain

The analysed process involved 10 cycles of temperature:

$$T = \begin{cases} 
293.15 - 8(t) & t \in [0h, 5h) \\
253.15(t) & t \in [5h, 10h) \\
253.15 + 8(t - 10) & t \in [10h, 15h) \\
293.15(t) & t \in [15h, 20h] 
\end{cases}$$

(10)

thus the total simulation time was equal to 200 h.

The frost induced degradation of the material microstructure, causing further increase of crystallization pressure, Fig. 1, and progressing material degradation during next freezing/thawing cycles, Fig. 2, is an important feature of frost damage, which should be considered in modelling of the process.

5. Conclusions

A mathematical model of hydro-thermal phenomena in a fully water saturated porous material, considering freezing/thawing phase changes by means of kinetic, non-equilibrium approach, and frost induced damage, is presented and numerically solved.

References

FE analyses of a coupled energetic-statistical size effect in concrete beams under bending

Ewelina Korol\textsuperscript{1}, Jacek Tejchman\textsuperscript{2}\textsuperscript{*}
\textsuperscript{1,2}Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: esyroka@pg.gda.pl, tejchmk@pg.gda.pl

Abstract

The numerical FE investigations of a coupled deterministic-statistical size effect in unnotched concrete beams of similar geometry under quasi-static three point bending were performed within elasto-plasticity with non-local softening. The FE analyses were carried out with four different beam sizes. Deterministic calculations were performed with the uniform distribution of a tensile strength. In statistical calculations spatially correlated random fields were used to describe the material randomness. Calculations were made with various length of spatial correlation and coefficient of variation. The effect of a simultaneously varying, cross-correlated concrete material parameters was investigated.

Keywords: beam, concrete, cross-correlation, random fields, size effect

1. Introduction

A size effect phenomenon (nominal strength varies with a characteristic size of a structural member) is an inherent property of the behavior of many engineering materials. In the case of concrete materials, both the nominal structural strength and material brittleness (ratio between the energy consumed during the loading process after and before the stress–strain peak) always decrease with increasing element size under tension [1]. When the size and slenderness of the structure are relatively high and the fracture energy is relatively low, the global structural response is brittle. Thus, concrete becomes ductile on a small scale and perfectly brittle on a sufficiently large scale. The results from laboratory tests which are scaled versions of the actual structures cannot be directly transferred to them.

Two size effects are of a major importance in quasi-brittle and brittle materials: energetic (or deterministic) and statistical (or stochastic). According to Bazant and Planas [1] the deterministic size effect is caused by the formation of a region of intense strain localization with a certain volume (micro-crack region called also fracture process zone FPZ) which precedes macro-cracks. The nominal structural strength which is sensitive to the size of FPZ cannot be appropriately estimated in laboratory tests, since it differs for various specimen sizes. Strain localization volume is not negligible to the cross-section dimensions and is large enough to cause significant stress redistribution in the structure and associated energy release. The specimen strength increases with increasing ratio $l_c/D$ ($l_c$ – characteristic length of the microstructure influencing both the size and spacing of localized zones, $D$ – characteristic structure size). In turn, a statistical (stochastic) effect is caused by the spatial variability/randomness of the local material strength.

2. Size effect laws

Two size effects laws proposed by Bazant ([1],[2]) (called size effect laws SEL) for geometrically similar structures allow for determining their nominal strength by taking into account the size scale effect. Type I applies to structures of a positive geometry having no notches or pre-existing cracks for which the maximum load occurs as soon as the FPZ is fully developed and the macroscopic crack can initiate. Type II occurs also for structure with a positive geometry but with notches or large stress-free cracks that grow in a stable manner up to the maximum load. The structures obeying the size effect of Type I are sensitive to the material randomness. The following energetic (deterministic) formula SEL of Type I predicted by asymptotic matching was proposed by Bazant ([1],[2])

\[
\sigma_c(D) = f_c \left(1 + \frac{D_n}{D + l_c}\right)^\frac{1}{2}
\]  

(1)

A simplified formula for a coupled energetic-statistical size effect law reads ([2]):

\[
\sigma_s(D) = f_c \left(\frac{D_n}{D}\right)^\frac{1}{2} + \frac{rD_n}{D + l_c}\]

(2)

where $m$ is the dimensionless Weibull modulus (shape parameter of Weibull distribution) responsible for the slope of a large-size asymptote and $n$ is the number of spatial dimensions in which the structure is scaled ($n = 2$). The empirical parameter $D_0$ drives the transition from elastic-brittle to quasi-brittle. Eq.(2) satisfies three asymptotic conditions: (a) for small structures $D \to 0$, it asymptotically reaches the plastic limit (Eq.(1)), (b) for large sizes $D \to \infty$, it asymptotically reaches the dominating Weibull size effect with the slope equal to $-n/m$ and (c) for $m \to \infty$, it is equal to the deterministic size effect law. Thus, Eq. (2) can be regarded as the asymptotic matching of small-size energetic and large-size statistical size effects.

3. FE input data

Our extensive numerical FE investigations of a coupled energetic-statistical size effect in unnotched concrete beams of a similar geometry under quasi-static three point bending were performed within elasto-plasticity with non-local softening to...
obtain mesh-independent results [3]. The FE analyses were carried out with four different beam sizes scaled in two dimensions. The deterministic calculations were performed with the uniform distribution of the tensile strength \( f_t \). In order to reduce the number of stochastic calculations, the stratified sampling technique was applied [3]. Initially, the FE calculations were performed with the spatially varying tensile strength \( f_t \) only, the length of the spatial correlation \( l_{cor} \) ranged from 10 mm up to 140 mm and the variation coefficient \( \text{cov} = \frac{\sigma_{ft}}{\mu_{ft}} \) (keeping the constant \( \mu_{ft} \)) varied between 0.12 and 0.20. In the next FE calculations, the simultaneously varying tensile strength \( f_t \), fracture energy \( G_f \) and modulus of elasticity \( E \) were assumed.

4. FE results

The influence of the varying correlation length \( l_{cor} \) was investigated based on small \((D = 80\, \text{mm})\), medium \((D = 160\, \text{mm})\), large \((D = 160\, \text{mm})\) and very large size beam \((D = 1920\, \text{mm})\). The mean tensile strength was \( \mu_{ft} = 3.6 \, \text{MPa} \) while the material coefficient of variation \( \text{cov} = 0.12 \) \((\text{cov} = \frac{\sigma_{ft}}{\mu_{ft}})\). 12 FE simulations were performed (using Stratified Sampling method) for each beam size and correlation length. The correlation length \( l_{cor} \) changed from 10 mm up to 140 mm. The reduction of \( F_{exp} \) with reference to deterministic value is stronger when the beam size \( D \) increase. The relation between \( l_{cor} \) and \( F_{exp} \) was not monotone. First the value of \( F_{exp} (l_{cor}) \) decrease and next increase. The coefficient of variation \( \text{cov} \) has first upward trend and eventually stabilized in case of the small, medium and large size beam. Results show that the influence of \( l_{cor} \) on \( F_{exp} \) and \( \text{cov} \) changes depending on the beam size. This behaviour is connected with the size of the localized zone at the peak load. Figure 1 presents SEL curves (Eq.2) fitted to stochastic numerical results with various \( l_{cor} = 10-120 \, \text{mm} \).

The influence of the material coefficient of variation \( \text{c.o.v.} = \frac{\sigma_{ft}}{\mu_{ft}} \) (assuming constant mean tensile strength \( \mu_{ft} \)) was investigated with \( \text{c.o.v.} = 0.12, 0.16 \) and 0.20. Four beam sizes (small, medium, large and very large) were considered. The correlation length \( l_{cor} = 80 \, \text{mm} \) was constant for all beam sizes. Fig.2 presents calibrated SEL curves (Eq.2) compared to obtained stochastic results with various \( \text{c.o.v.} = 0.12-0.20 \). The statistical size effect on the nominal flexural strength \( f_t \) becomes stronger with the beam size increase. The influence of the material coefficient of variation \( \text{c.o.v.} \) on the statistical size effect is pronounced. The higher \( \text{c.o.v.} \) the strongest statistical size effect.

References

Two-dimensional FE analysis of confined concrete column

Artur Kotarski¹, Zdzisław Więckowski²

¹,²Faculty of Civil, Architectural and Environmental Engineering, Łódź University of Technology
Al. Politechniki 6, 90-924 Łódź, Poland
e-mail: Artur.Kotarski@p.lodz.pl, Zdzislaw.Wiecekowski@p.lodz.pl

Abstract

The stiffness and strength of a confined concrete column is investigated in the paper by means of the finite element method. As in the case of the axial compression state, the assumption that the cross-section of the column remains plane and perpendicular to the longitudinal axis is valid, the strain state of the column can be regarded as the generalised plane strain state with a given longitudinal contraction. This allows to analyse the equilibrium problem for the column by the use of a two-dimensional finite element computer code. The column constituents: concrete and reinforcement are considered elastic–plastic. Numerical results have been compared with the experimental ones available in literature.

Keywords: finite element method, elastic–plastic material model, spiral reinforced column, confine concrete

1. Introduction

The behaviour of a confined concrete column has been investigated mostly in experimental studies and rather less intensely by the use of numerical methods like the finite element method, e.g. [1, 2]. In the present paper, the finite element method—as in [1, 2]—is utilized in the analysis of the problem. However, in contrast to the papers cited above where the three-dimensional problem has been considered, the two-dimensional approach is proposed here. In the case of compression of a prismatic column, the stress state in a cross-section of the column can be considered independent of a position of the section along the column. This means that the stress field is a solution of the generalised plane strain problem given contraction of the column acting as a load. An advantage of such approach is a much smaller effort in preparing data for a two-dimensional analysis than that in the case of the three-dimensional calculations described in [1, 2]. A simple constitutive model of an elastic–perfectly plastic body is used to model the constituents of a column. In the analysis, the influence of the cross-section part representing concrete outside spirals is neglected as spalling is expected when column’s compression is advanced. Several types of a column’s cross-section have been analysed. The method allows one to model easily a column’s cross-section of arbitrary shape with various patterns of lateral reinforcement like spirals or stirrups.

2. Setting of the problem

Let us consider a prismatic concrete column with a uniformly distributed confining reinforcement. When $x_3$-axis is oriented along the column, the strain state is described by three components $\varepsilon_{\alpha\beta}$ ($\alpha, \beta = 1, 2$) to be found with one given $\varepsilon_{33} = -E_{33}$ where $E_{33}$ is the longitudinal shortening of the column. The last two strain components, $\varepsilon_{3\alpha}$ ($\alpha = 1, 2$) are equal to zero. Four components of the stress tensor are to be found: $\sigma_{\alpha\beta}$ and $\sigma_{33}$ ($\sigma_{33} = 0$). The main purpose of the analysis is to find the axial force in the column,

$$F = -\int_{\Omega} \sigma_{33} \, dx_1 \, dx_2 - A_l \, \sigma$$  \hspace{1cm} (1)

where $\Omega$ denotes the cross-sectional region, as a function of shortening $E_{33}$. In Equation (1), $A_l$ and $\sigma$ are the area of the cross-section of longitudinal reinforcement and the reinforcement stress, respectively.

A simple elastic–perfectly plastic constitutive model with the Drucker–Prager yield condition and a non-associated flow rule is utilized to describe the behaviour of concrete. The constitutive relations for concrete can be written in the following form:

$$\dot{\varepsilon}_{ij} = \dot{\varepsilon}_{ij}^e + \dot{\varepsilon}_{ij}^p, \quad \dot{\varepsilon}_{ij}^p = C_{ijkl} \dot{\sigma}_{kl},$$

$$\dot{\varepsilon}_{ij}^p = \begin{cases} \frac{\dot{\lambda}}{\tau} & \text{if } f(\sigma_{ij}) = 0 \quad \text{with } \dot{\lambda} \geq 0, \\ \dot{\lambda} & \text{if } f(\sigma_{ij}) < 0, \end{cases}$$  \hspace{1cm} (2)

where a dot denotes the time derivative, $\dot{\varepsilon}^e$ and $\dot{\varepsilon}^p$ are the elastic and plastic components of the strain tensor, respectively, $C_{ijkl}$ denotes the tensor of elastic compliance, and $f$ the plastic yield function,

$$f = q - m \, p - k \quad \text{with } \quad m = \frac{6 \sin \varphi}{3 + \sin \varphi}, \quad k = \frac{6 \cos \varphi}{3 + \sin \varphi}.$$  

In the above definition, $\varphi$ is the angle of internal friction, $c$ the cohesion while $p$ and $q$ are the following invariants of the stress tensor $\sigma_{ij}$ and deviatoric stress tensor, $s_{ij}$,

$$p = \frac{1}{3} \sigma_{ii}, \quad q = \sqrt{\frac{3}{2} s_{ij} s_{ij}} \quad \text{where } s_{ij} = \sigma_{ij} + p \, \delta_{ij}.$$  

In Equation (2), $g$ is the non-associated plastic flow potential satisfying relation $g = q$.

3. Finite element solution

As a computational tool for the analysis, the finite element method has been chosen, the problem is formulated in the variational form using the equation of virtual work

$$\int_{\Omega} \sigma_{ij} \delta \varepsilon_{ij} \, d\Omega + \int_{S} A \sigma \delta \varepsilon \, ds = 0 \quad \forall \delta \mathbf{u} = 0 \in V_0$$  \hspace{1cm} (3)

where $S$ is a line representing the axis of the compression, $A$ the area of a reinforcement cross-section, $\sigma$ and $\varepsilon$ denote the normal
stress and longitudinal strain in the reinforcement, $\delta u$ a variation of the displacement field. The latter belongs to the space of kinematically admissible displacement fields, $V_0$, that are sufficiently regular and satisfy the homogeneous kinematic boundary conditions.

After using the finite element interpolation for the displacement field by means of constant strain triangular elements for region $\Omega$ and 2-node bar elements for the reinforcement, Eq. (3) takes the following form:

$$\int_\Omega B^T \sigma \, d\Omega + \int_S A B^T \sigma \, ds = 0$$

where $B$ and $\bar{B}$ are the stress–displacement matrices for the plane strain triangular and bar elements, respectively, and $\sigma$ denotes the stress vector, $[\sigma_{11} \quad \sigma_{22} \quad \sigma_{12}]^T$. As in both the cases of plane strain and bar elements, the linear interpolation functions are used, the condition of continuity of displacement field is satisfied on the concrete–reinforcement interface.

The equation system (4) is solved in an incremental way by increasing the magnitude of longitudinal strain component $\varepsilon_{33}$ equivalent to column contraction taken with the negative value, $-E_{33}$. The contraction plays a role of loading. For each load increment, the system of non-linear equation (4) is solved iteratively by means of the modified Newton–Raphson method. The implicit procedure ([4, 6]) is employed to calculate the stresses satisfying the constitutive relations (2).

4. Example

The proposed computational model was applied in the analysis of several cross-sections of a confined concrete column like rectangular and square sections with multi-spiral reinforcement and a square cross-section with a simple stirrup. The results obtained for the square section with five spirals are demonstrated in the paper. The column cross-section and the space discretisation of a quarter of the computational region are shown in Figure 1.

![Figure 1: Cross-section of analysed column and triangulation of computational region](image)

Computations have been made with the following material data for concrete: Young’s modulus 32 GPa, Poisson’s ratio 0.2, compressive strength 30 MPa, and for steel: Young’s modulus 210 GPa and yield limit 274.7 MPa. To apply the Drucker–Prager plasticity model, the value of the internal friction angle has been set, $\phi = 37^\circ$ as indicated in [3]. The cohesion value has been chosen as $c = 8.551$ MPa to fit the Drucker–Prager yield surface to the compressive meridian of the Mohr–Coulomb surface.

The results of the computations are shown in the form of relation between the contraction of the column and the averaged compressive stress in the cross-section in Figure 2. The left plot presents the relation obtained in the case of the transversal reinforcement ratio equal to 2.24 % with the main spiral thinner than the corner ones. The right plot is related to the reinforcement ratio of 2.20 % with all the spirals of the same thickness.

![Figure 2: Averaged stress—longitudinal strain relation](image)

The obtained results are compared with the experimental ones shown in [5]. The differences are rather small, particularly, in the second case of the reinforcement. In both the presented cases, the computed stresses are smaller than those observed in experiments. One of the reasons of underestimation of the values may be the fact that the hardening of steel has not been included in the analysis.

5. Conclusions

A two-dimensional finite element model has been proposed for an analysis of the stress state in the cross-section of a confined concrete column. As the stress in the column can be regarded as independent of the location of the section along the column, the problem has been formulated as a generalisation of the plane strain problem. A rather simple elastic–plastic constitutive model has been applied for both the constituents of the column: concrete and steel reinforcement. Several shapes of a column cross-section with spiral and stirrup reinforcement have been analysed. The finite element results have been compared with the outcomes of experiments. A good agreement between both the results has been obtained.

References

Investigations of vortex and anti-vortex structures in sand
during plane strain compression by DEM

Jan Kozicki¹, Jacek Tejchman²

¹,² Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: jkozicki@pg.gda.pl ¹, tejchmk@pg.gda.pl ²

Abstract

The paper presents selected simulation results of a quasi-static plane strain compression test on cohesionless sand under constant lateral pressure using a three-dimensional discrete element method (DEM). Grains were modelled by means of spheres with contact moments imitating irregular particle shapes. Sand was initially dense. Material behaviour was studied at both global and local levels. The stress-strain and volumetric-strain curves, distribution of void ratio, resultant grain rotation and contact forces were obtained. The main attention was paid to the appearance of micro-structures like vortex and anti-vortex structures in the specimen.

Keywords: plane strain compression test, granular material, discrete element method, vortex, anti-vortex, shear localization

1. Introduction

Vortex structures are frequently observed in experiments on granular materials - they are manifest directly grain rearrangement and therefore they can be thought of as a basic mechanism of irreversible deformation [1,2,3,4]. They became apparent when the motion associated with uniform strain is subtracted from the total particle motion. They are reminiscent of turbulence in fluid dynamics, however the amount of the cells rotation is several ranges of magnitude smaller (~0.01°-0.1°) than the fluid vortex rotation. The vortex structures were mainly observed in shear zones which are fundamental phenomena observed in granular soils under drained and undrained conditions. The objective of the paper is to report the results of comprehensive studies by a discrete element method DEM analysis of the vortex structures during quasi-static drained plane strain compression of the cohesionless sand specimen 4×14×8 cm³ by taking shear localization into account. In order to accelerate the computation time, some simplifications were assumed: large spherical elements with contact moments, linear sphere distribution, linear normal contact model and no particle breakage. A three-dimensional discrete model YADE developed at University of Grenoble was applied [5]. The discrete calculations were solely carried out with initially dense sand for one confining pressure and initial void ratio.

2. Numerical results

In order to precisely detect all vortices based on the displacement fluctuation map, a special method used in the physics was applied, based on the orientation angles of the displacement fluctuation vectors of the neighbouring single spheres [6]. Initially the displacement fluctuation field of the irregularly distributed spheres was turned into a regular orthogonal lattice by computing the mean displacement fluctuations of spheres inside the cubic cell 5d₅₀×5d₅₀×5d₅₀ at the points spaced by the 0.5d₅₀-distance in the both directions on the 2D grid (d₅₀=2.5 mm - the mean grain diameter of sand). To find the centre of a vortex, a group of 4 angles (spins), which were the corners of the square 0.5d₅₀×0.5d₅₀ on the 2D grid was chosen. Each corner was the result of the average displacement fluctuation from the cube 5d₅₀×5d₅₀×5d₅₀. The set of 4 spins was chosen to determine whether they rotated by ±2π as the eye went from one spin to the next in a clockwise or an anti-clockwise direction around the square (Fig.1). The vortex was detected in the lattice when the spins of the normalized displacement fluctuation vectors rotated by at least Σα=-2π, i.e. as the eye moved clockwise around the closed path (Fig.1a) or Σα=2π, i.e. in the anti-clockwise direction (Fig.1b). Note that the vector length was ignored. In turn, the anti-vortex (equivalent with shear, Fig.1d) might occur if Σα=−2π as the eye moved in an anti-clockwise direction or Σα=2π as the eye moved in a clockwise direction (it could also be left- and right-handed). In order to eliminate the frequency noise (caused by the fact that spins rapidly and chaotically changed their directions as the eye moved to the next one), the maximum allowed angle which varied between the neighbouring corners was taken as 160° (Fig.8a). If any angle between two neighbours was larger than 160°, the check in the cell was aborted. The approach was also applied with a higher number of vectors around the selected point 'p' - e.g. with 8 vectors (Figs.1c and 1d) that allowed to determine more vortices.

The vortex and anti-vortex structures solely occurred in the shear zone (Fig.2). They appeared and disappeared surprisingly during the entire deformation process. First, the vortices and anti-vortices developed in the region of a future forthcoming shear zone. Based on their formation, the next two intersecting shear zones occurred at the vertical normal strain ε₁=0.6% and then one dominant inclined shear zone formed at ε₁=1.5%. The maximum temporary number of the vortices was 13 at ε₁=11% and ε₁=25.5% and of the anti-vortices was also 13 at ε₁=16% and ε₁=26% along the shear zone. Their maximum temporary number which was concentrated at one place was 7. At the beginning of deformation (ε₁<1.5%), the more anti-vortices happened, then the vortices (1.5%≤ε₁<5%) dominated. Next, these micro-structures appeared always in the dominant inclined shear zone. The right-handed vortices and left-handed anti-vortices happened more frequently than the left and right-handed ones due to the shear direction of the granular specimen. The right-handed vortices seemed to be connected with the left-handed anti-vortices and the left-handed vortices with the right-handed anti-vortices.
Their number in Fig. 2 was the following: 6 anti-vortices occurred in the mid-point of two intersecting shear zones at $\varepsilon = 1\%$, 6 vortices and 7 anti-vortices appeared at 3 different locations in one inclined shear zone $\varepsilon = 3\%$, 6 vortices occurred in the mid-point of the shear zone (at $\varepsilon = 5\%$), 6 vortices and 5 anti-vortices at 2 different locations in the shear zone (at $\varepsilon = 10\%$), 3 vortices and 6 anti-vortices at 2 different locations (at $\varepsilon = 15\%$), 3 vortices and 2 anti-vortices at two different locations of the shear zone (at $\varepsilon = 20\%$), 4 vortices at one position of the shear zone (at $\varepsilon = 25\%$) and 5 vortices and 5 anti-vortices at 2 different locations in the shear zone (at $\varepsilon = 30\%$).

It was observed that maximum 6 concentration regions of the vortices and anti-vortices might simultaneously appear along the shear zone line at $\varepsilon = 8.6\%$, 11.1\%, 14\%, 18.9\%, 20.4\%, 21\% and 23.8\%. Their distance was different, varying between 0.1$l_{sh}$ and 0.8$l_{sh}$ ($l_{sh}$ - the shear zone length). The predominant period of the right-handed vortices was 12\% of $\varepsilon_1$, of the left-handed vortices 4\% of $\varepsilon_1$, of the right-handed anti-vortices 9\% of $\varepsilon_1$ and of the left-handed anti-vortices 15\% of $\varepsilon_1$.

3. Conclusions

The method used for the detection of all vortex structures was very effective. The vortex and anti-vortex structures solely occurred in the shear zone or in the region of the shear zone. They appeared and disappeared during the entire deformation process. They happened in both directions. The right-handed vortices and left-handed anti-vortices were more frequent than the left-handed vortices and right-handed anti-vortices, respectively.

The vortices and anti-vortices turned out to be a precursor of shear localization.

References

Micromechanics of hydrating cement pastes considering progressive C-S-H gel densification

Markus Königsberger¹, Bernhard Pichler², Christian Hellmich³

¹,²,³ Institute for Mechanics of Materials and Structures, Vienna University of Technology (TU Wien)
Karlsplatz 13/202, A-1040 Vienna, Austria

Abstract

A novel poromicromechanical model for hydrating Portland cement pastes is presented. The heterogeneous material is represented by a three-scale approach, resolving the porosity (gel and capillary pores) on two different scales. Special attention is given to calcium-silicate-hydrates (C-S-H), the glue in cementitious materials. We consider a hydration degree-dependent morphology of these crystals as well as the their anisotropic stiffness characteristics. Evolutions of phase volume fractions stem from a novel hydration model. It accounts for densification of the C-S-H gel, as evidenced by NMR relaxometry experiments. Based on a recent experimental micromechanics approach in the framework of the eigenstress influence tensor concept, homogenization of macroscopic material behavior is performed. Model-predicted drained and undrained stiffnesses match experimental results nicely.

Keywords: cement paste, micromechanics, poromechanics, densification, C-S-H, gel porosity, multiscale modeling

1. Introduction

After water, cementitious materials like mortar or concrete are the most used materials on the planet. Given large amounts of carbon dioxide released during cement production, great efforts are recently made to optimize the mechanical behavior in order to reduce the ecological footprint of these materials. The aforementioned optimization process calls for a careful study of the components building up cementitious material, from the nanoscale up to the macroscale.

We focus on cement paste, a complex hierarchically organized heterogeneous material [5, 15], built up by hydrates (mainly Calcium-Silicate-Hydrate, C-S-H) which stem from the chemical reaction of water and cement clinker. Using a continuum micromechanics approach allows us to resolve material heterogeneity on several scales of observation. We are interested in predicting the poroelastic material behavior of hydrating cement pastes. Thereby, we consider recent experimental information on the densification behavior of the C-S-H hydrates [8], a gelatinous composite of solid C-S-H crystals and gel pores. This way, we are able and predict mechanical material behavior as function of the degree of the chemical reaction between water and cement clinker. Experimental validation is carried out on the macroscopic scale of cement pastes.

2. Space confinement-driven C-S-H gel densification

Nuclear magnetic resonance (NMR) relaxometry and Rietveld analyses of X-ray diffraction patterns [8] have shown that C-S-H gel densities progressively during hydration. Small angle scattering experiments provide insight into the density and the composition of C-S-H crystals [2]. Combining both information, under consideration that space confinement is the driving force for C-S-H gel densification, generalize the measured densification behavior for pastes exhibiting arbitrary composition in terms of the water-to-cement mass ratio w/c and the degree of the chemical reaction \( \xi \), see [6] for details. Basic stoichiometric relations of hydration chemistry together with a simplified hydration kinetics (following the famous Powers-Acker hydration model [1, 13]) finally results in an original hydration model for the evolution of phase volume fractions of hydrating Portland cement pastes, as function of \( w/c \) and of \( \xi \). In contrast to Powers model, our approach explicitly accounts for nonlinear hydrate volume increase during cement clinker consumption, as evidenced by NMR [8].

3. Multiscale microporomechanics model

Inspired by existing succesful micromechanics approaches [3, 11, 12, 14], a multiscale poromechanics model involving three observation scales is developed for hydrating cement pastes, see Fig. 1, with microstructural features described as follows. At the scale of several nanometers, solid C-S-H crystals are represented as oblate ellipsoids, evidenced by experimental methods as NMR [16] and small angle neutron/X-ray scattering [4]. Moreover, aspect ratios of the crystals are considered functions of the hydration degree \( \xi \). In more detail, the earlier the crystals precipitate, the closer is their shape to plates; the later they precipitate, the closer is their shape to spheres. Solid C-S-H crystals, intermixed with nanometer-sized and water-filled gel pores, form the C-S-H gel at the scale of one micron. At this scale of observation, a porous polycrystal called C-S-H foam is introduced. It consists of a highly disordered mixture of C-S-H gel needles with capillary pores. The C-S-H foam together with embedded spherical clinker grains and Portlandite crystals finally represents cement paste.

Homogenization is based on (i) the described hierarchical representation of cement paste, (ii) the novel hydration model accounting for C-S-H gel densification, and (iii) a continuum micromechanics approach developed in the framework of the eigenstrain influence tensor concept [10]. In more detail, starting from the stiffness tensor of solid C-S-H crystals, recently identified by means of atomistic modeling [7, 9], we upscale poroelastic properties to the material scale of hydrating cement pastes. We predict drained and undrained stiffnesses of cement pastes for any given combination of the composition parameters \( w/c \) and \( \xi \). Comparing the drained model-predicted Young’s moduli to mechanical test results as well as the undrained model-predicted Young’s and shear moduli to dynamic quantities from ultrasonic experiments, shows a very good agreement between predictions and test results. In particular, drained predictions match the experimentally determined almost linear stiffness increase during hydration very well.
4. Conclusions and outlook

Using a homogenization approach incorporating the complex microstructure of cement paste as well as C-S-H gel densification characteristics, we predict the macroscopic poroelastic behavior of hydrating Portland cement pastes. Successful experimental validation at the cement paste scale strongly corroborates our multiscale model developments.

As a next step, it will be interesting to quantify stress concentrations, a prerequisite for strength predictions. Moreover, our model opens the door to studying time-dependent phenomena of cementitious materials such as creep.

References


Investigation of micro-macro relationships of elastic parameters in the discrete element model of granular material

Izabela Marczewska¹, Jerzy Rojek², Rimantas Kacianauskas³*

¹,²Institute of Fundamental Technological Research, Polish Academy of Sciences
Pawińskiego 5B, 02-106 Warsaw, Poland
e-mail: imar@ippt.pan.pl, jrojek@ippt.pan.pl²
³Vilnius Gediminas Technical University
Sauletekio al. 11, 10223 Vilnius, Lithuania
e-mail: rimantas.kacianauskas@vgtu.lt

Abstract

A general objective of the paper is to improve understanding of micromechanical mechanisms in granular materials and their representation in numerical models. Results of numerical investigation on micro-macro relationships in the discrete element model of granular material are presented. The macroscopic response was analysed in a series of simulations of the triaxial compression test. Numerical studies were focused on the influence of microscopic parameters on the elastic response. The effect of the contact stiffness and the contact stiffness ratio on the effective elastic moduli, the Young’s modulus and Poisson’s ratio, were investigated. Numerical results were compared with the analytical estimations.

Keywords: discrete element method, granular material, triaxial test, micro–macro relationship, Voigt hypothesis, elastic moduli

1. Introduction

The discrete element method (DEM) employing spherical particles became a very popular framework to model granular materials. In the DEM, a material is represented as a large collection of rigid particles (discrete elements) interacting with one another by contact forces. The discrete element model belongs to a class of micromechanical material models. Calibration of the discrete element models aiming to establish the relationship between microscopic and macroscopic model parameters is the key issue in an effective use of the discrete models to model real materials. Constitutive micro-macro relationships for discrete element models of granular materials can be obtained analytically or numerically.

In the present work, numerical simulations of the triaxial compression test will be performed aiming to establish the relationships between the microscopic parameters and macroscopic properties of a granular material. The investigation is focused on the initial response and elastic effective moduli of a granular material. Numerical results will be compared with analytical micro-macro relationships derived from the Voigt kinematic hypothesis.

2. Analytical micro-macro relationships

The Voigt hypothesis is based on the assumption that a particle assembly is subjected to a uniform strain state and the displacements of individual particles are in accordance with the displacement field induced by the uniform strain.

For an assembly of spherical particles of the same size and same material properties with isotropic packing structure, the closed-form formulae for the equivalent macroscopic Young’s modulus $E$ and Poisson’s ratio $\nu$ can be derived in the following form [2, 3]:

$$E = \frac{n_c(1 - \epsilon)k_n}{2\pi r}, \quad \frac{2k_n + 3k_t}{4k_n + k_t}, \quad \nu = \frac{k_n - k_t}{4k_n + k_t} \tag{1}$$

where $n_c$ is the so-called coordination number, a parameter defined as an average number of contacts per particle, $k_n$ and $k_t$ are the contact stiffness in the normal and tangential direction, respectively, $r$ is the particle radius, and $\epsilon$ is the specimen porosity. The Voigt kinematic hypothesis leads to an upper bound solution for elastic moduli. It should be reminded that the above expressions are valid for the stick contact without sliding and without any moment-type interaction at the contact point.

3. Discrete element simulation results

Numerical simulations of the triaxial compression test were performed using a discrete element code DEMpack [1] for the cylindrical specimen shown in Fig. 1. The specimen is composed of 5,430 spherical particles with radii varying from $r_{min} = 0.462$ mm to $r_{max} = 0.89$ mm. The initial porosity of the specimen is $\epsilon = 0.39$. The particle mass density is equal to $\rho = 1600$ kg/m$^3$.

![Figure 1: Discrete element model of the triaxial compression tests](image-url)
The granular material is modelled using the cohesionless contact model with sliding friction, neglecting a rolling resistance. The interparticle Coulomb friction coefficient $\mu = 0.5$ was assumed. The confining rigid walls were assumed nearly smoothly with the Coulomb coefficient $\mu = 0.02$ for the particle–wall friction. The problem was studied for different values of micromechanical parameters: the contact stiffness $k_n$, and the ratio of the contact stiffness in the tangential and normal directions $k_t/k_n$.

The results for different values of the contact stiffness $k_n$, the constant ratio $k_t/k_n = 0.35$ and the confining pressure 100 kPa are shown in Figs. 2 and 3 in the form of the curves showing the evolution of the deviatoric stress and volumetric strain, respectively, as functions of the axial strain.

$$2\pi Er/(n_c(1-e)k_n) = 2 + 3(k_t/k_n) + 4 + (k_t/k_n)\frac{1}{4 + (k_t/k_n)}$$

$$\nu = \frac{1 - (k_t/k_n)}{4 + (k_t/k_n)}$$

The right-hand sides of Eqs. 2 provide theoretical predictions plotted in Figs. 4 and 5. The dimensionless parameters involving the Young’s modulus $E$ and Poisson’s ratio $\nu$ were determined from the initial slopes of the curves shown in Figs. 2 and 3. Dimensionless relationships involving the macroscopic elastic moduli and microscopic parameters $k_t$ and $k_n$ are plotted in Figs. 4 and 5. The dimensionless parameters involving the Young’s modulus $E$ and Poisson’s ratio $\nu$ are obtained by rewriting the equations (1) as follows:

$$2\pi Er/(n_c(1-e)k_n) = 2 + 3(k_t/k_n) + 4 + (k_t/k_n)\frac{1}{4 + (k_t/k_n)}$$

$$\nu = \frac{1 - (k_t/k_n)}{4 + (k_t/k_n)}$$

The right-hand sides of Eqs. 2 provide theoretical predictions plotted in Figs. 4 and 5.

4. Conclusions

It can be seen that the numerical results are bounded by the theoretical predictions derived using the Voigt hypothesis. The lower is the contact stiffness $k_n$, the closer are the numerical results to the theoretical ones.

The analytical formulae for macroscopic elastic moduli are based on the assumption that there is no sliding at contacts, therefore the difference between the numerical and theoretical results is smaller for small values of the contact stiffness -- the smaller the stiffness the bigger is the penetration of the particles and interparticle sliding and rearrangement play a minor role in an overall particle assembly deformation.

References


Description of damage process in sedimentary rocks

S. Pietruszczak1, E. Haghighat2
1,2 Department of Civil Engineering, McMaster University, Hamilton, Ont., Canada L9S 4L7
e-mail: pietrusz@mcmaster.ca, haghige@mcmaster.ca

Abstract

The paper is focused on the analysis of the deformation process in sedimentary rocks that display a strong inherent anisotropy. The constitutive relation is formulated within the plasticity framework and deals with both homogeneous deformation mode, prior to failure, and the localized deformation. The effects of anisotropy are accounted for by incorporating the notion of a microstructure tensor. The crack propagation process is modelled through a discrete representation of a constitutive law with embedded discontinuity (CLED). The framework is applied to examine the onset and propagation of damage around a tunnel excavated in a deep sedimentary rock formation.

Keywords: anisotropy, microstructure tensor, strain localization, discrete crack propagation, embedded discontinuity

1. Introduction

Argillaceous sedimentary rocks are typically formed by deposition and progressive consolidation (including diageneric processes) of marine sediments. They exhibit a strong degree of inherent anisotropy (transverse isotropy), so that their stiffness and strength properties are directionally dependent. The anisotropy is due to the oriented microstructure, particularly the presence of bedding planes that can be easily seen by a visual inspection. The specific type of rock considered here is the Tournemire shale/argillite. This formation is located in a Mesozoic marine basin on the south limit of the French Massif Central.

The mechanical properties of Tournemire have been examined by a number of researchers. An example here are the laboratory investigations of Niandou et al. [1]. In addition to a number of hydrostatic compression tests, the authors performed a series of conventional triaxial tests on cylindrical samples (37 mm in diameter and 75 mm in height). The tests were carried out at different confining pressures and different orientations of the bedding planes and included unloading-reloading cycles. Strains in three orthogonal directions (i.e. longitudinal, parallel and perpendicular to the bedding planes) were measured. The results generally indicate that the maximum strength is associated with specimens in which the direction of major principal stress is either parallel or perpendicular to the bedding planes, while the minimum strength was observed for orientations between 30 and 60 degrees. The failure mode and the degree of anisotropy both evolve with the confining pressure. At high pressures, the response is ductile while at low confinement the strain softening behaviour, associated with brittle failure, takes place. In general, the stress-strain characteristics show a significant non-linearity and irreversibility of deformation. After unloading, approx. 50% of the total deformation is in the plastic range for both longitudinal and transversal directions. There is also a strong experimental evidence that the behaviour of Tournemire shale is time-dependent and the effect of creep, particularly at higher deviatoric stress intensities, is very significant.

The paper is an extension of the work reported earlier [2] and deals with description of the deformation process in sedimentary rocks. Both, homogeneous and localized modes are considered here, while the influence of anisotropy is incorporated by invoking the microstructure tensor approach. The strain localization is assumed associated with formation of macrocracks and a simple methodology is discussed for identifying the orientation of the crack based on the critical plane approach. In the following, the plasticity framework incorporating an anisotropic deviatoric hardening model is outlined first and the formulation of a constitutive model with embedded discontinuity [3] is reviewed. The integration scheme incorporates a closest-point projection algorithm, which is employed for both stages of the anisotropic deformation process. The paper is concluded by providing a numerical example, which deals with modeling of localized damage around a deep tunnel excavated in Tournemire argillite. The crack path is monitored in a discrete manner by using the level-set method.

2. Methodology

The description of the mechanical behaviour requires, first of all, the specification of conditions at failure under an arbitrary stress state. In addition, a general framework must be provided for the evaluation of the deformation field, which may include discontinuities such as macrocracks. Over the last few decades, an extensive research effort has been devoted to modeling of the mechanical behaviour of anisotropic rocks. A comprehensive review on this topic examining different approaches is provided, for example, in the article by Duveau and Henry [4]. The approach employed here incorporates anisotropy measures which depend on relative orientation of principal axes of stress and microstructure tensor (after Ref. [5]). Those descriptors are directly identified with strength parameters, so that the strength properties are assumed to be orientation-dependent. The plasticity formulation is derived by assuming the yield/loading surface in the functional form

\[
\begin{align*}
\phi &= \bm{\sigma} - \bm{\sigma}_0 = 0; \\
\theta &= \bm{\eta}_f \frac{\kappa}{J_3 + \kappa}; \\

\end{align*}
\]

(1)

where \(\bm{\sigma}_0 = (J_3) \frac{\mu}{J_3}, \quad \sigma_m = -\frac{I_1}{3}\) and \(\theta\) is the Lode’s angle. The hardening effects are attributed to accumulated plastic dis-
tortion $\kappa$, while the function $g(\theta)$ is assumed in the form consistent with Mohr-Coulomb criterion. Note that the latter is obtained by setting $\theta \rightarrow \eta_f$ in Eqn (1).

The effects of anisotropy are accounted for by assuming that the strength descriptor $\eta_f$ is orientation dependent, i.e.

$$\eta_f = \hat{\eta}_f (1 + A_g l_i l_i + b_1 (A_g l_i l_i)^3 + b_2 (A_g l_i l_i)^3 + \ldots )$$

(2)

Here $A_g$ is a symmetric traceless operator whose eigenvectors define the principal material axes, while $l_i$ is a so-called loading vector whose components are the normalized magnitudes of traction vectors the planes normal to preferred axes. Note that the parameter $C$ is associated with a hydrostatic stress state and it remains invariant with respect to orientation of the sample. The flow rule is non-associated.

For the localized deformation mode, the orientation of the macrocrack is assessed by invoking the critical plane approach. The damage propagation process is then modeled through an enhanced discrete representation of the constitutive law with embedded discontinuity [3]. For the discontinuous motion, the symmetric part of the velocity gradient is defined as

$$V^\times \mathbf{v}(x,t) = V^\times \mathbf{v}(x,t) + \mathcal{H}(\varphi) V^\times \mathbf{v}(x,t) + \delta(\varphi) (\mathbf{n} \otimes \mathbf{v})$$

(3)

Here, $\mathbf{v}$ and $\mathbf{v}$ are two continuous functions, $\mathcal{H}(\varphi)$ is the Heaviside step function and $\delta(\varphi)$ is the Dirac delta function and is assumed in the form

$$\delta(\varphi) = \delta(\varphi)(\mathbf{n} \otimes \mathbf{v})$$

Here, $\mathbf{v}$ and $\mathbf{v}$ are two continuous functions, $\mathcal{H}(\varphi)$ is the Heaviside step function and $\delta(\varphi)$ is the Dirac delta function.

In general, the specification of material constants requires information on the conditions at failure as well as the deformation characteristics in samples tested at different orientation of the bedding planes and different confining pressures. Here, the results of a series of axial compression tests were employed to identify the function $\eta_f(l_i)$ and the material parameter $C$, Eqs. (1-2). In particular, for each specific orientation of the sample and the given confining pressure, the value of $\eta_f$ and the corresponding loading direction were determined.

The formulation was employed to examine the onset and propagation of discrete damage around underground openings in the Tournemire shale formation. It was demonstrated that the localized shear band/macrocracking mechanism can noticeably change the deformation/stress field around the opening. This is of importance, particularly when investigating structures that may pose serious safety hazards, e.g. Deep Geologic Repositories for the long-term management of radioactive wastes.

References


3. Numerical analysis

As mentioned in section 1, the numerical study deals with the Tournemire shale. The study makes use of the results of triaxial tests that have been reported in Ref. [6]. Those results were employed to identify the material parameters/functions that enter the formulation, as discussed in section 2.
Explanation of the mechanism of destruction of the cylindrical sample in the Brazilian test

Jerzy Podgórski¹, Jakub Gontarz²*

¹² Faculty of Civil Engineering and Architecture, Lublin University of Technology
Nadbystrzycka 40, 20-618 Lublin, Poland
e-mail: j.podgorski@pollub.pl ¹, j.gontarz@pollub.pl ²

Abstract

The paper presents the analysis of the so-called Brazilian compression tests of the cylinder loaded by two linearly distributed balanced forces, in terms of possibility of determining the proper tensile strength of the material. These analyses contain the precisely determined stress field, without singularity at the point of force application, the determination of the critical stress from the point of view of the classical and contemporary failure criteria for brittle materials, and the position of the point at which the destructive crack may start to destroy the sample. The classical mechanism of destruction is based on the analysis of plane stress which requires revision and its replacement with a 3D model. These analyses are supported by experimental data from laboratory tests of the authors.

Keywords: Brazilian test, concrete mechanics, rock mechanics, failure criteria, material effort ratio

1. Introduction

The paper presents the study of the indirect method of determining the tensile strength of brittle materials such as concrete and rock. Most often this type of testing is done using the "Brazilian test" by compressing the cylindrical sample by the two linear, balanced loads. The simplicity of this test, and the convenience of using drilling cores as laboratory samples made the “Brazilian test" the dominant method for determining the tensile strength of natural rock and concrete. The tensile strength ($f_t$) of the sample material is usually determined by taking the maximum tensile stress ($\sigma_t$), reached at the moment of destruction. The stress value is usually determined by means of a 2D elasticity problem solution for the circular shield compressed by two balancing forces acting along a diameter, which gives:

$$f_t = \frac{2P_{\text{max}}}{\pi d h}, \quad (1)$$

where $P_{\text{max}}$ is the force destroying the sample, $d$ – diameter and $h$ – the height of the cylindrical sample.

![Figure 1: Problem of the circular disk compression - "Brazilian test". a) loaded by a concentrated force, b) distributed load, c) distributed load considered in FEM analysis](image)

Figure 1: Problem of the circular disk compression - "Brazilian test", a) loaded by a concentrated force, b) distributed load considered by Hondros, c) distributed load considered in FEM analysis

Tensile strength, determined this way, is lower than the value determined by a direct tensile test. The reason is an excessive simplification in determining the stress field, neglecting the compressive stress effect, which have a significant impact on material effort.

The problem of determining the tensile strength of the material on the basis of the measurement results during the Brazilian test is still an interesting topic of research, which can be seen in many works appearing in scientific journals devoted to the problems of rock mechanics and concrete [1, 2, 4]. The authors have attempted to accurately determine the tensile strength using the Brazilian test on the basis of the material effort analysis calculated with the modern and classical failure criteria applicable to concrete and natural rocks. Conditions of Lame-Rankine, Coulomb-Mohr, Drucker-Prager, Ottosen-Podgórski were considered.

2. Stress field

Determination of the stress field in the circular disc compressed along the diameter (Fig 1a) is a classic problem, solved in the late nineteenth century by Flemish and Hertz.

At the point of the concentrated force application, the stress field singularities are observed because $\sigma_x \to -\infty$ and $\sigma_y \to \infty$.

In the case of the real problem of the cylinder loaded by distributed pressure, the stress field singularities do not exist, because the pressure is always applied to the small area of the cylinder surface (Fig. 1b). In this case, the stress field has been designated by Hondros [4]. In our analyses of sample destruction, the stress field was obtained by numerical methods (FEM), for the problem which corresponded to the cylinder loaded by pressure distributed on a portion of the cylinder surface (Fig. 1c).

3. Criteria for material damage

In the paper impact of the selection of failure criterion on the location of the largest material effort was analyzed. Failure criteria proposed by Lame-Rankine, Coulomb-Mohr, Drucker-Prager and Ottosen-Podgórski were applied.

The Ottosen-Podgórski condition [5] has proposed in the form of relation of three alternative stress tensor invariants:

$$\sigma_0 - C_0 + C_1 P_J \tau_0 + C_2 \tau_0^2 = 0, \quad (2)$$

*This research was sponsored by the statutory fund of the Faculty of Civil Engineering and Architecture, Lublin University of Technology.
where \( P(J) \) is a function describing the cross-section of limit state surface by the deviatoric plane, proposed by Podgórski in the form:

\[
P(J) = \cos \left( \frac{1}{2} \arccos(\alpha J) - \beta \right) .
\]

where \( \alpha, \beta, C_0, C_1, C_2 \) are constants dependent on the material.

Comparing the ordinate of the intersection point of the load path in the Brazilian test (\( \sigma_1 = \kappa \sigma_2 \)) with the envelope of the failure criterion (Fig. 2, 3), it can be seen that only in the case of the simplest Lame-Rankine criterion, the maximum tensile stress is obtained in the Brazilian test regarded as tensile strength \( f_t = \sigma_{\text{max}} \), in the other cases \( f_t > \sigma_{\text{max}} \) holds.

Due to other criteria, receiving a simple relationship between \( \sigma_{\text{max}} \) and \( f_t \) is not possible, yet small differences between envelopes Drucker-Prager or JP conditions can be observed, and the parabola of the equation:

\[
\sigma_2 = -\frac{1}{2} f_t \pm \sqrt{\left(\frac{1}{2} f_t\right)^2 + \frac{\sigma_1 f_t}{f_t}}
\]

can be assumed with sufficient accuracy so that the point of intersection of the parabola with load path for the Brazilian test gives the required value: \( \sigma_{\text{max}} = \rho f_t \), where:

\[
\rho = \frac{\sqrt{1 + 4 \gamma^2} - 1}{2 \gamma^2}, \quad \gamma = \frac{\kappa}{\eta}, \quad \eta = \frac{f_c}{f_t}
\]

Taking \( \kappa = 3.0 \) and \( \eta = 10 \), we receive \( \rho = 0.92328 \). Tensile strength calculated using JP and Drucker-Prager criterion is therefore approx. 8% higher than the \( \sigma_{\text{max}} \), i.e.

\[
f_t = \frac{\sigma_{\text{max}}}{\rho} = 1.083 \sigma_{\text{max}}.
\]

As a criterion for initiation of destructive crack, the achieved maximum material effort was taken. Material effort (\( \mu \)) is defined as the ratio of the vector modules:

\[
\mu = \frac{r_i}{r_f},
\]

where \( r_i \) is a vector indicating a point (in stress space) corresponding to the stresses in the sample site, and \( r_f \) is a vector indicating a point on the failure criterion surface which also belongs to the monotonic load path (Fig. 4).

Location of a point initiation the destructive crack is an interesting issue too. As shown in the graph (Fig. 4b), the place of maximum effort depends on the ratio \( \eta = f_c / f_t \). When \( \eta < 17 \), the initiation point of destruction moves toward the loaded edge of the sample and at the \( \eta \geq 17 \), the point of maximum effort is located in the center of the sample (Fig. 4b).

The real mechanism of destruction of the sample is different and its explanation requires taking into account the spatial work of the sample material. The corresponding analysis performed by finite element method taking into account the JP fracture criterion and triaxial stress state, explains the mechanism of destruction and indicates the origin of the destructive crack (Fig. 5).

References


Dynamic behaviour of Zymne Monastery Cathedral on soil base with consideration of non-linear deformation of materials.

Vladimir Sakharov
Base and Foundation Department, Kyiv National University of Construction and Architecture
Povitroflotskiy av., 31, 03680, Kyiv, Ukraine
e-mail: vladland@gmail.com

Abstract

The study contains results of numerical analysis of the seismic impact on massive constructions considering Zymne Monastery Cathedral building as an example. The cathedral has been founded in X-XI century AD using mostly brickwork. Due to the numerous destructions, reconstructions, change of ownership through history between Poland, Soviet Union and Ukraine, the cathedral has a complex configuration, in the immediate need of strengthening its foundation. Simulation has been performed using FEM with modified explicit central difference method in the nonlinear formulation. The calculations involved a soil model that takes into account visco-elastic-plastic deformation properties, structural strength and water pressure in soil pores. The study also contains an analysis of the influence of nonlinear orthotropic properties of the brick masonry on the stress-strain state of the building and the estimation of the dynamic behaviour of the cathedral structures. It demonstrates the distribution of stress concentration zones and locations of the localized structural damage. Finally, it contains an analysis of the accumulated plastic deformations in the soil base that leads to an uneven foundation settlement and the estimations how the water pressure in the soil pores affected the considered processes.

Keywords: nonlinear soil, pore pressure, brick masonry, orthotropic properties, seismic, direct explicit method, ASSR“VESNA-DYN”

1. Introduction

Due to the increase of the seismic activity in the Ukraine, architecturally and historically valuable buildings need to be inspected for seismic resistance. During this process it is important to consider not only the specific soil conditions and material parameters, but also the real nonlinear properties of the environment.

2. Interaction of the Cathedral building with a soil base

2.1. Problem definition

The goal is to ensure reliable operation of buildings and structures under dynamic load, including seismic activity. Due to the recent general increase of the seismic activities in Ukraine, it is necessary to review a seismic resistance of existing buildings and structures. About 12% of Ukrainian territories are currently classified seismically dangerous areas [1]. A numerical simulation has been performed in order to evaluate stress-strain state (SSS) of a massive, predominantly brick-based cathedral building under the seismic load. Numerous studies have shown large influence of the irreversible deformation of the soil mass and structures on the dynamic behaviour of the structures [2].

2.2. Structural features of the Cathedral

This study presents results of the analysis of the massive building based on the monument of architecture that belongs to the XI-XII century AD - Uspenskij Cathedral of the Zymne Monastery in Volyn, Ukraine. Founded by the Vladimir The Great, it experienced numerous destructions, reconstructions and architectural changes over its more than 1000-year long history, all of which affected its present state. The main load-bearing structures of the cathedral, including its foundation are made using brick masonry, laid before the 1495 AD. The Cathedral is located on the slope with a retaining wall built in the upper part of the slope.

The geological structure consists of layers of loam and sandy loam, underlain by Cretaceous sediments. On the surface, natural soil is covered with filled ground with depths from 0.5 to 5.0 m. Seismicity of the territory is considered to be 6 points by MSK-64 scale.

2.3. Finite element model

The simulation was performed using actual geographic relief, enclosing structures and the strengthening elements of the foundation. The study was conducted using a developed finite element model (FEM), which included a soil array, cathedral building and the retaining wall as a “soil base - foundation - building” system (Fig. 1).

Figure 1: Finite element model of the Cathedral as a "soil base - foundation - building" system

The soil base and the cathedral were modelled using volumetric finite elements utilizing FEM moment scheme [3,4], which resulted in improved convergence. The final model is...
described by a system of algebraic equations with about one million variables. Modelling of the seismic action was carried out on the basis of an automated system for Scientific Research (ASSR) «VESNA-DYN» in spatial statement by a modified method with the explicit nonlinear deformation of materials [5].

3. Environment models

During simulation, soil mass was modelled by a nonlinear visco-elastic-plastic body, while masonry model took into account nonlinear orthotropic elastic-plastic properties.

3.1. Soil model

A model used to describe soil behaviour takes into account the structural strength, visco-elastic-plastic volume, shape deformation processes [2,6] and processes associated with the pore pressure. Thus simulation was possible of real processes of interaction between the soil base and the foundation under seismic loads. A structural and rheological model of soil is shown in Fig. 2.

![Figure 2: Structural-rheological model of visco-elastic-plastic deformation of the soil under dynamic load](image)

The model is based on the ideas of the Kelvin viscoelastic model of three branches. The first branch shows an element H0, representing an elastic modulus, the element S of the plastic strength and the viscous element Ns of the creep process, supplemented with a brittle element P of structural strength. The second branch simulates a high-speed of viscoelastic properties of the soil. Finally, third branch is responsible for the simulation of the pore pressure and water filtration process. In addition a unilateral constraint R simulates features of ground tension. The model also takes into account viscous energy dissipation from the interaction with the environment.

3.2. Brick masonry model

Modelling the deformation of brick masonry walls and piers of the Cathedral was based on a piecewise linear stress-strain dependency diagram (Fig. 3). Orthotropic properties appear during the deformation and accumulation of local damage along the respective directions. Orthotropic axes are oriented in accordance with the masonry joints. At the initial stage the masonry is considered an isotropic material.

![Figure 3: Stress-strain dependency graph for the masonry](image)

If the tensile strength of brick masonry is exceeded, the cracks are formed in a plane perpendicular to the direction of the stress while corresponding elastic modulus and Poisson's ratio tend to zero. The moduli of elasticity in other directions remain unchanged. There is also a possibility that cracks will close in the future – then the compressive strength of masonry is restored and only shear modulus is corrected. During the formation of cracks in two directions it is assumed that the masonry material is crushed, so it will not be able to sustain any load.

4. Results

The simulation has shown that the nonlinear properties have a significant impact on the structural behaviour of the cathedral. They identified localized damage zones caused by the tension and compression, predominantly located under squinches, at the base of pylons and buttresses and at several places near the entrance. Analysis also revealed the formation of the non-uniform sediment, causing tilt of the structure. The pore pressure in the water-saturated layer increased the calculated foundation settlement values. However, it should be noted that the discovered damaged zones are not sufficient to fundamentally affect work of the load-bearing structures of the cathedral and will not cause emergency situation.

References

Modelling of concrete fracture at aggregate level using FEM and DEM based on real microstructure

Łukasz Skarżyński¹, Michal Nitka², Jacek Tejchman¹*
¹ Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: lska@pg.gda.pl ¹, mienitka@pg.gda.pl ², tejchmk@pg.gda.pl ³

Abstract

The paper describes two-dimensional numerical results of fracture at aggregate level in a notched concrete beam under quasi-static three-point bending. Two different numerical approaches: continuum and discrete were used. Concrete was modelled as a random heterogeneous 4-phase material composed of aggregate particles, cement matrix, interfacial transitional zones (ITZs) and air voids. Within continuum mechanics, the simulations were carried out with the finite element method (FEM) based on a isotropic damage constitutive model enhanced by a characteristic length of micro-structure by means of a non-local theory. As a discrete approach, the discrete element method (DEM) was used. The concrete micro-structure in calculations was directly taken from a real concrete specimen based on 3D by x-ray micro-tomography images and 2D images by the scanning electron microscope (SEM). Attention was paid to the shape of a fracture zone.

Keywords: concrete, discrete element method (DEM), finite element method (FEM), fracture, non-local theory, micro-structure, X-ray micro-CT, interfacial transitional zones (ITZs)

1. Introduction

Fracture is a fundamental phenomenon in concrete materials [1,2]. It is very complex since it consists of main cracks with various branches, secondary cracks and micro-cracks [1]. During fracture, micro-cracks first arise in a hardening region on the stress-strain curve which change gradually during material softening into dominant distinct macroscopic cracks up to damage. The fracture process strongly depends upon a heterogeneous structure of materials over many different length scales, changing e.g. in concrete from the few nanometres (hydrated cement) to the millimetres (aggregate particles). In order to properly describe fracture in detail, material micro-structure has to be taken into account since its effect on the global results is pronounced [2]. The understanding of a fracture process is of major importance to ensure the safety of the structure and to optimize the material behaviour.

At the meso-scale, concrete may be considered as a composite material wherein four important phases may be separated: cement matrix, aggregate, interfacial transition zones ITZs and macro-voids. In particular, the presence of aggregate and ITZs is important since the volume fraction of aggregate can be as high as 70-75% in concrete and ITZs with the thickness of about 50 µm are always the weakest regions in common concrete wherein cracking starts (because of their higher porosity).

The main objective of this paper is to investigate fracture out under two-dimensional (2D) conditions in concrete elements under bending at aggregate level using 2 different approaches: a meso-continuum (FEM) [3] and discrete (DEM) one [4]. Concrete was considered as a four-phase body composed of aggregate, cement matrix, ITZs and air voids. The concrete micro-structure was directly taken from real concrete specimen using non-destructive techniques, based on 3D x-ray micro-tomography images and 2D scanning electron microscope (SEM) images. The FE simulations of tensile deformation were carried out with a simple isotropic continuum constitutive damage model enhanced by a characteristic length of micro-structure by means of a non-local theory [3]. The DEM calculations [4] were performed with the three-dimensional spherical discrete element model YADE, which was developed at the University of Grenoble [5]. The numerical outcomes were directly compared with the experimental results. Attention was paid on the crack shape and macroscopic force-deflection response.

2. Experimental results

The concrete was prepared from the ordinary Portland cement (CEM I 32.5 R), aggregate and water. The mean aggregate diameter was $d_{ss}=2$ mm, maximum aggregate diameter $d_{max}=16$ mm and aggregate volume of $\beta=75\%$. The test was carried out on simply supported rectangular notched concrete beam (depth $D=80$ mm, width $B=40$ mm and length $L=320$ mm). The notch of depth of $D/10=8$ mm and width of 3 mm was located in the beam mid-span. The quasi-static beam test were performed with a controlled notch opening displacement rate (crack mouth opening displacement (CMOD)) of 0.002 mm/min using the loading machine Instron 5569. After bending test, two concrete cuboidal specimens with the dimensions of 80 mm (height), 50 mm (width) and 40 mm (depth) was cut out from the beams in the notch region for scanning by the x-ray micro-tomograph Skyscan 1173 in order to obtain the 3D images of concrete micro-structure (in a damaged state). The test and calculations were performed for quasi-static conditions.

In FEM, a simple isotropic damage continuum model was used which describes the material degradation with the aid of only a single scalar damage parameter $D$ growing monotonically from zero (undamaged material) to one (completely damaged material) [2,3]. For DEM the 3D spherical discrete element model YADE was chosen [5] with a linear normal contact model under compression [4].

Concrete modelled by as a random four-phase heterogeneous material with angularly-shaped aggregate particles in the region close to the notch. The width of the meso-region was equal to 50 mm, equal to the width of the cuboidal specimens. The location, shape, size and distribution of aggregate grains and air voids directly corresponded to the

¹*The research work has been carried out within the project: "Experimental and numerical analysis of coupled deterministic-statistical size effect in brittle materials" financed by National Research Centre NCN (UMO-2013/09/B/ST8/03598).
concrete images from X-ray. The air voids (with the diameter ≥0.8 mm and the total surface of approximately 2.5%) were modelled as the spots.

In FEM aggregate was solely described by a linear elastic model. The width of ITZs was constant 0.05 mm. The calculations were carried out under plane stress conditions by taking the width of the beam into account h=40 mm when calculating the vertical force. The following parameters were assigned to each individual phase: aggregate (Emacro=47.2 GPa, ν=0.2), cement matrix (Ecm=29.2 GPa, ν=0.2, TITZ=1.5×10^4, α=0.8, β=200), ITZ (EITZ=14.6 GPa, ν=0.2, κ,ITZ=1×10^4, α=0.95, β=200), where κ is damage parameter, α and β are the material constants. In the remaining region, the material was described as the elastic one-phase material (Emacro=36.1 MPa and Tmacro=0.2).

In DEM, aggregate grains were modelled as clusters composed of spheres with the diameter of d = 0.5 mm connected to each other as rigid bodies. All aggregate grains included the ITZs. The cement matrix was modelled with the spheres with the diameter d=0.25-2 mm without ITZs. The remaining beam region (outside the meso-region close to the notch) was simulated with the spheres of d=2-8 mm. In order to significantly shorten the computations, the beam included only one layer of grains [4]. Based on preliminary calculations of uniaxial compression and uniaxial tension tests [4], the following parameters of cohesion and tensile strength were assigned to the cement matrix (Ecm=29.2 GPa, Ccm=140 MPa and Tcm=25 MPa) and ITZs (EITZ=20.4 GPa, CITZ=100 MPa and TITZ=17.5 MPa), where C is a shear cohesion and T is a normal cohesion. In the remaining region outside the meso-region with large grains was described by the constants: Emacro=36.1 MPa, Cmacro=140 MPa and Tmacro=25 MPa. The contact elastic stiffness of the cement matrix and beam macrozone were taken directly from laboratory tests (i.e. they were the same as in the FEM calculations). The remaining parameters were constant for all phases and regions: v=0.2 (Poisson’s ratio of grain contact), μ=18° (inter-particle friction angle), α=0.08 (damping parameter) and ρ=2.6 kg/m³ (mass density).

Figures 1 demonstrate the force-CMOD curves obtained for a concrete beam with the real aggregate and air voids distribution in three different cross-sections in the notch region using FEM and DEM as compared to the experiments. The calculated (FEM) and experimental force-CMOD curves are similar in the entire hardening-softening regime. The calculated maximum vertical force was F=2.31-2.37 kN for CMOD=0.017-0.23 mm in 3 different vertical cross-sections. In the experiments, the maximum vertical force was F=2.25 kN for CMOD=0.016 mm. The residual vertical forces were also similar in the calculations and experiments.

In the DEM computations the agreement of the curve F=f(CMOD) with the experiments was slightly worse (curve 'c'). The calculated maximum vertical force was F=2.22-2.70 kN for CMOD=0.015-0.016 mm. The calculated residual forces were higher.

In the experiments, the main crack was strongly curved mainly due to presence of aggregate grains (Fig. 2a). Its shape changed along the specimen width despite the fact that 2D boundary value problem (plane stress) was considered. The crack mainly propagated through ITZs (which were the weakest phase in concrete) and sometimes through macro-voids. It might very rarely propagate through a single weak aggregate particle. The crack branching also occurred. The effect of air macro-voids on the crack shape was small. Numerical results of fracture were similar with the experiments (Figs.2b and 2c).

3. Conclusions

The calculation 2D results at aggregate level using an isotropic damage continuum model enhanced by a characteristic length of micro-structure and discrete element model satisfactorily captures the evolution of fracture above the beam notch under quasi-static three-point bending, considering concrete a heterogeneous four-phase material.

The experimental crack above the notch is strongly curved since it propagates through the weakest contact. The existence of ITZs has a pronounced influence on the material strength and fracture.

References

Numerical homogenisation of permeability coefficient for Darcy flow in porous media

Marek Wojciechowski

Faculty of Civil Engineering, Architecture and Environmental Engineering, Technical University of Lodz
Al. Politechniki 6, 90-924 Łódź, Poland
e-mail: mwoj@p.lodz.pl

Abstract

Novel numerical method is used for homogenisation of permeability coefficient in porous media. Flow of water, at both macro and micro level, is assumed to be ruled by Darcy law. A special averaging constraint is applied at microscale, which allows to apply macroscopic pressure gradient without the necessity to use directly Dirichlet or Neumann boundary conditions. This approach allows arbitrarily shaped representative volumes and eliminates undesirable “boundary effects” which can violate solution. We exploited this concept in this paper and we show that it is useful for permeability homogenisation.

Keywords: numerical homogenisation, boundary conditions, Darcy flow

1. Introduction

In a broad class of disordered porous materials with clear matrix-inclusions internal structure, the flow of fluid can be described by Darcy equation at both, micro and macro level. This is for example the case of concrete and sand-clay soils. In order to determine the macroscopic response with accurate account for microstructural characteristics and evolution, computational homogenization strategy can be explored [3]. Recently a new concept of representative volume element (RVE) analysis has been proposed [4], which consist in applying special averaging constraint to the microscopic (RVE) analysis has been proposed [4], which consist in applying special averaging constraint to the microscopic problem, instead of Dirichlet or Neumann boundary conditions. This allows for arbitrary shapes of RVE, and eliminates undesirable “boundary effects” which can violate solution. We exploited this concept in this paper and we show that it is useful for permeability homogenisation.

2. Homogenisation framework

Let’s consider microstructurally complex porous material for which a representative volume element (RVE) of the volume \(\Omega\) be defined. In case of laminar flow, local flux \(\mathbf{u}_i\) in the RVE can be given by Darcy law (skipping source terms):

\[
\mathbf{u}_i = -k_{ij} \frac{1}{\gamma} p_j
\]

where \(p_j\) is a pressure gradient, \(k_{ij}\) is a permeability tensor depending on the position in the RVE (in velocity units) and \(\gamma\) is specific weight of the fluid. Averages of the microscopic fluxes and pressure gradients over domain \(\Omega\) are given by:

\[
P_j = \frac{1}{\Omega} \int_{\Omega} p_j d\Omega
\]

\[
U_i = \frac{1}{\Omega} \int_{\Omega} u_i d\Omega
\]

These values are assumed to be related by the effective permeability \(K_{ij}\), such that:

\[
U_i = -\frac{K_{ij}}{\gamma} p_j
\]

what describes the macroscopic behavior of the composite.

The homogenization problem considered here is formulated as follows: find solution \(p\) to equations (1) defined on \(\Omega\), subject to some macroscopic pressure gradient \(P_j\) such a way that equation (2) is fulfilled. From this solution microscopic \(U_i\) and macroscopic \(U_i\) and \(K_{ij}\) are then derived.

The above can be viewed as a problem of minimization of total potential energy (see [2]) with additional averaging constraint, which can be applied by Lagrange multipliers \(\lambda_k\):

\[
\min_p \left[ \int_{\Omega} -p_j \frac{k_{ij}}{\gamma} p_j d\Omega + \lambda_k \left( -\Omega P_k + \int_{\partial\Omega} p_j d\Omega \right) \right]
\]

3. Finite element method formulation

The minimization problem (5) can be rewritten in the scope of finite element method as:

\[
\min_p [p^T Ap + \lambda^T (-\Omega P + B^T p)] = \min_p [\Pi(p, \lambda)]
\]

where \(p\) is global vector of unknown pressures of the length \(M\) (\(M\) - total number of nodes in discretization), \(A\) is a global linear operator of the size \(M \times M\), \(B\) is a problem specific matrix of the size \(M \times D\) (\(D\) - space dimension: 2 or 3), \(\lambda\) is a vector of unknown Lagrange multipliers of the length \(D\), and \(P\) is a vector of known, macroscopic pressure gradient to be applied, also of the length \(D\). Solution to the problem is found by differentiation of the potential \(\Pi\) defined by equation (6) with respect to unknown \(p\) and \(\lambda\) and equating the results to 0:

\[
\frac{\partial \Pi(p, \lambda)}{\partial p} = Ap + B\lambda = 0
\]

\[
\frac{\partial \Pi(p, \lambda)}{\partial \lambda} = B^T p - \Omega P = 0
\]

This is a system of linear equations:

\[
\begin{bmatrix}
A & B^T \\
0 & \Omega
\end{bmatrix}
\begin{bmatrix}
p \\
\lambda
\end{bmatrix} =
\begin{bmatrix}
0 \\
0
\end{bmatrix}
\]

For this problem only a minimal set of boundary conditions should be applied, without introducing any additional pressure gradient and flux. This is achieved simply fixing pressure at certain level: \(p = p_0\) at single, arbitrary boundary point of the domain (or at arbitrary boundary node of its discretization).
Macrosopic, effective permeability can be now computed from the observation, that Lagrange multipliers $\lambda$ are interpreted as macroscopic average flux, with minus sign, i.e.: $\lambda = -U$. Combining equations (7) and (8) we get then:

$$\nabla P = (B^T A^{-1} B) U$$
$$U = \Omega (B^T A^{-1} B)^{-1} P$$

From equations (10) and (4) we get the effective permeability:

$$K_{ij} = K = -\gamma\Omega (B^T A^{-1} B)^{-1}$$

(11)

Note that equation (11) does not need the actually explicit solution of the system of linear equations (9).

4. Numerical example

The presented homogenization method for a Darcy flow was implemented in the frame of finite element package fempy [5]. We performed calculations for a two-dimensional representative volume of the sand-clay mixture [1]. RVE has reference size about 5 mm and irregular boundary adjusted to representative volume of the sand-clay mixture [1]. RVE has volume ratio of sand grains and void ratio of clay paste. We performed calculations for a two-dimensional triangular finite elements. A finest discretization (10647 nodes, 21003 elements) is used for better handling grain shapes, but also coarser mesh should provide acceptable results (it was not tested though). In the example the reference pressure was taken as 0 kPa at single boundary point of the discretization, and the macroscopic pressure gradient of the value: $P_j = [1, 1]$ kPa/m is applied.

$$U_{ij} = \frac{1}{10} \frac{1}{10}$$

(12)

It is straightforward to verify that equation (4) holds for these results.

Macroscopic permeability obtained in this numerical example is generally anisotropic, but it is almost equal in both Cartesian directions, i.e. $K = K_{11} \approx K_{22} \approx 3.7 \cdot 10^{-10}$ [m/s] and the skew component tends to be a small value. This indicates that the randomly generated RVE is appropriate for the considered problem – statistically isotropic at macroscale. Comparisons to the oedometric laboratory tests reported in [1] show that computed permeability $K$ fall, as expected, into the range of variation of experimental results performed for similar volume ratio of sand grains and void ratio of clay paste.

5. Conclusions

Numerical homogenization approach is an attractive method for dealing with Darcy flow in heterogeneous porous media. This is because of the possibility of taking into account local proportions, arrangements, shapes and permeability parameters of composite constituents. At every integration point of macroscopic problem such locally defined representative volume elements are “loaded” with local macroscopic pressure gradients, in an average sense, to get the macroscopic flux vectors. No special shape of RVE is needed for the presented method. The structuring effects, i.e. irregular, elongated paths of water flow through the RVE, visible in Fig. 1, are automatically taken into account. This is not the case in analytical approaches like stratified or self-consistent homogenisation methods.

References


Incorporation of crack closure effect in damage-plasticity models

Adam Wosatto\(^1\), Aikaterini Genikomou\(^2\), Jerzy Pamin\(^3\), Maria Anna Polak\(^4\), Andrzej Winnicki\(^5\)
\(^{1,3,5}\) Faculty of Civil Engineering, Cracow University of Technology
Warszawska 24, 31-155 Cracow, Poland
e-mail: awosatto@is.pk.edu.pl
\(^{1,2,4}\) Department of Civil and Environmental Engineering, University of Waterloo
200 University Avenue West, Waterloo, Ontario, N2L 3G1, Canada

Abstract

Crack closing incorporation in the stress-strain relationship for a two damage-plasticity models is presented. Gradient-enhanced damage can be coupled to plasticity and in this model averaging equation prevents the pathological discretization sensitivity. Concrete damaged plasticity is available in ABAQUS and can be equipped with viscoplastic regularization. In both models it is possible to activate crack closure effect. Basic features of models and representative examples in the context of usefulness for numerical analysis of crack closure phenomenon are briefly discussed.

Keywords: damage, plasticity, crack closing, stiffness recovery, concrete, regularized models

1. Introduction

The majority of quasi-brittle materials during the experiments reveal the stiffness recovery for reversed cyclic (pseudo-seismic) loading, cf. e.g. tests for RC slab-column connections [1] or masonry-infilled RC frames [3]. This effect is also called crack closing. The constitutive model in the nonlinear analysis should be able to reproduce a proper stiffness in each phase of the stress evolution. For example, while compression follows tension and microcracks and microvoids close in the body, the elastic behavior under compression and the initial stiffness should be at least partly retrieved. In the paper two nonlinear and nonlocal models including the crack closure phenomenon are briefly discussed.

2. Scalar damage coupled to plasticity

2.1. Concise description of model

The gradient damage theory formulated in the strain space is coupled with the plasticity theory formulated in the space of effective stresses [2]. The damage measure \(\omega\) is scalar and grows from 0 to 1. The damage growth function depends on damage history parameter \(\kappa\). The postulate of strain equivalence [7] holds, so the fictitious counterpart represents the undamaged “skeleton” of the body and the effective stress tensor \(\tilde{\sigma}\) is related to the real stress tensor \(\sigma\). The damage loading function \(f^d\) is defined using equivalent strain measure \(\tilde{\epsilon}\).

For the Burzyński-Drucker-Prager plasticity theory linear isotropic hardening is assumed. Equivalent strain measure \(\tilde{\epsilon}\) can be a function of total strain \(\epsilon\), so also plastic strain \(\epsilon^p\) can contribute to damage function \(f^d\). However, in the employed version of the model the coupling of damage with plasticity is weaker and only the elastic part of strain \(\epsilon^e\) influences damage.

It is possible to enhance the model incorporating the crack closure phenomenon in the constitutive relation. The tensile part \(\epsilon^{e+}\) is separated from elastic strain \(\epsilon^e\) using a projection operator \(P^+\) [10]. The positive (tensile) strains are responsible for damage. If crack closing is active then the tangent stress-strain relationship is formulated in the rate form:

\[
\sigma = E_{\text{tan}}\tilde{\epsilon} - \tilde{\sigma}^+ \tilde{\omega}
\]  

where \(E_{\text{tan}}\) is the tangent stiffness operator including effects of damage-plasticity coupling and the strain projection for crack closing. \(\tilde{\sigma}^+\) is the tensile part of the effective stress tensor. If crack closing is omitted then operator \(E_{\text{tan}}\) is reduced to \((1 - \omega)E^0\) and \(\tilde{\sigma}^+\) becomes \(\tilde{\sigma}\). \(E^0\) is the elasto-plastic tensor.

The finite element formulation is derived from weak-forms of the equilibrium equation and an additional averaging equation:

\[
\tilde{\epsilon} - c\nabla^2\tilde{\epsilon} = \tilde{\epsilon}_1 - \tilde{\epsilon}_2
\]

in which the \(\nabla^2\) operator acts on the averaged strain \(\tilde{\epsilon}\) [6]. The parameter \(c > 0\) has a unit of length squared and it is connected with the internal length scale \(l\) of a material. The presence of this diffusion-type equation in the gradient-enhanced model ensures that the issue of spurious discretization sensitivity is overcome in localization simulations. In the finite element implementation the averaged strain measure is discretized in addition to the displacements.

Further details and other aspects of the model are described in [5, 11]. The finite elements are programmed in the FEAP package [9].

2.2. Example

One finite element test with static tension in one horizontal direction and compression in the opposite direction is performed for a plane stress configuration. The difference between the incorporation of crack closing in the pure damage model and in the coupled damage-plasticity model is considered. Young’s modulus \(E\) equals 20000 MPa and Poisson’s ratio is zero. Linear softening is applied for the damage model. Damage threshold \(\kappa_o\) is 0.0001 and the ultimate value of history parameter \(\kappa_u\) is 0.004. For HHH plasticity yield strength \(\sigma_y = 2\) MPa and linear hardening with \(h = 2\) is adopted. The crack closure phenomenon is activated by means of the projection operator \(P^+\).

Figure 1 presents loading-unloading-reloading paths for stress \(\sigma_{11}\) versus strain \(\epsilon_{11}\). If damage without coupling with plasticity and without the projection operator is analyzed, the stiffness in the reloading is kept from the unloading, so the lack of the recovery is visible. Crack closing for pure damage is noticed when the point representing material stress and strain upon unloading returns to the origin in the diagram and the elastic stiff-
ness is restored. Figure 1 also depicts the crack closure effect in the coupled model. If plasticity is included then irreversible strain is obtained. Next, while the projection operator is employed, the initial stiffness is observed during the reloading process.

![Crack closing in damage coupled to plasticity](image1.png)

Figure 1: Crack closing in damage coupled to plasticity

### 3. Concrete damaged plasticity

#### 3.1. Concise description of model

The concrete damaged plasticity model, available in ABAQUS [8], was originally presented in [4]. The yield function is introduced in an effective stress space. Different one-dimensional relations are adopted for the evolving material strengths in tension and compression. Plastic potential flow similar to Burzyński-Drucker-Prager surface is employed and the determination of dilatancy effect on results is mandatory. Additionally, the stiffness degradation can be activated in different ways, hence two damage variables ωt and ωc are considered for tension and compression, respectively.

The crack closure phenomenon called here the stiffness recovery effect is enclosed in this model. The stiffness degradation depends on the stress state and the uniaxial damage variables. For the uniaxial cyclic conditions it is assumed:

\[ 1 - \omega = (1 - s_t \omega_t)(1 - s_c \omega_c) \]  

(3)

where \( s_t \) and \( s_c \) introduce stiffness recovery effects associated with stress reversals.

The model can be enriched with viscoplastic regularization according to a generalization of the Devaut-Lions approach, where viscous terms in plastic strain tensor and damage parameter are incorporated if additional time relaxation parameter \( \mu \) is larger than zero.

#### 3.2. Example

Similarly to the previous section the benchmark test is also performed as a simple verification of the model. Static unidirectional tension-compression is reproduced for 3D finite element configuration. Elasticity is defined by Young’s modulus \( E = 20000 \text{ MPa} \) and Poisson’s ratio \( \nu = 0.2 \). For plasticity the tensile strength \( f_t \) is equal to 2.0 MPa and next the postcracking behavior is applied as linear softening. When tensile cracking strain \( \epsilon^t_{cr} \) reaches 0.004 a constant residual tensile stress \( \sigma_t = 0.05 f_t \) is held. For compression linear-piecewise curve is introduced and the following values of inelastic strain \( \epsilon^m_{in} \) are adopted: 0.0, 0.0001, 0.0004, 0.0008, 0.002 together with corresponding values of compressive stress \( \sigma_c \): 5.0, 8.0, 9.0, 8.0, 5.0 [MPa]. The stiffness degradation is optional. Damage grows linearly to 0.8 when tensile cracking strain \( \epsilon^t_{cr} = 0.0005 \) and for compression when inelastic strain \( \epsilon^m_m = 0.002 \). The crack closure effect is also examined.

Three diagrams are depicted in Fig. 2 for one cycle of a reversal load. In case of plasticity the full elastic unloading is noticed, so the stiffness recovery cannot occur. While also damage is included in the model the change of the diagram in the unloading-reloading phase, the reduction of the stiffness and the lack of recovery are visible. The last diagram is performed for computations with activated crack closure effect. The damage unloading path agrees with the case "plasticity + damage" until reloading in compression when the elastic stiffness is restored and the slope agrees with the response for plasticity. In the compression regime when the sign of loading changes again, the stiffness differs in each case. For the crack closing the stiffness recovery in tension is also shown.

![Crack closing in concrete damaged plasticity](image2.png)

Figure 2: Crack closing in concrete damaged plasticity

### References


A model of stiffness of normal interaction of spherical particles embedded in matrix

Darius Zabulionis¹ Rimantas Kačianauskas², Vytautas Rimša¹, Jerzy Rojek¹

¹²³ Institute of Mechanics, Vilnius Gediminas Technical University, J. Basanavičiaus Str. 28, 03224 Vilnius, Lithuania
e-mail: darius.zabulionis@vgtu.lt ¹, rimantas.kacianauskas@vgtu.lt ², vytautas.rimsa@vgtu.lt ³
⁴ Institute of Fundamental Technological Research, Polish Academy of Sciences, Pawiriskiego ul. 3B, 02-106 Warszawa, Poland,
e-mail: jrojek@ippt.gov.pl

Abstract

A model of a normal interaction of deformable spherical particles embedded in matrix is examined analytically. Normal interaction was modelled by linearly elastic one dimensional springs transferring axial forces only. Two, upper and lower, limits of an axial stiffness of the spring for the normal interaction are proposed and validated using a 3D finite element analysis with different ratios of elastic moduli of particles and matrix and distances between particles. The validation showed that the results of FEM are between the proposed limits of stiffness of the normal interaction for a connecting element with particles stiffer than interface member.

Keywords: stiffness of normal interaction, spherical particles, particulate composite, heterogeneous materials

1. Introduction

An approximation of a medium by one dimensional bars is widely used in the modelling of various heterogeneous and composite materials: concrete, rocks, geomaterials, biomaterials, etc. [1,2,3,4]. Since a unified approach of bonded particles still does not exist various Lattice and/or Spring Network Models can be applied to model particulate composites.

This work is aimed at the assessment of the axial stiffness of a normal elastic interaction of spherical particles embedded in a weaker or stiffer matrix. Two upper and lower limits of the stiffness were achieved and verified by 3D FEM. An analysis showed that the results obtained by FEM are between the developed limits when matrix is weaker than particles and the FEM results are slightly bit greater than the obtained upper limit.

2. Modelling concept and governing equations

A particulate composite consisting of particles embedded in a matrix is approximated by one dimensional springs connecting centres of particles (Fig. 1). The springs are characterized by their length and axial stiffness $K_s$ only (Fig. 1b).

It is assumed that the particle and interface materials obey linearly elastic law, the connecting springs undertake only axial forces being normal interaction forces, particles of the composite do not rotate, particles interact with an interface member by an entire surface of hemispheres.

All parameters related to particles are denoted by subscripts $p$ while quantities related to interface members are denoted by a subscript $b$. The particle is characterised by radius $R_p$, elasticity modulus $E_p$ and Poisson’s ratio $v_p$. The interface member is characterised by the cylinder radius $R_b$ and the elasticity constants $E_b$ and $v_b$, respectively.

Two limits $K_{s,I}$ and $K_{s,II}$ of the axial stiffness $K_s$ can be obtained by considering different operations of division of a connecting element: in the form of parallel and series connected springs (Fig. 2).

Figure 1: A normal interaction of two spheres through a conditional interface member (a) and a spring representing an interaction of the spheres (b)

Figure 2: Concepts of the discretisation of a half of the conditional connecting members: by parallel connected prisms (a) and by sequentially connected rings (b)

For the two approaches, the total stiffnesses $K_{s,I}$ and $K_{s,II}$ of the connecting elements are as follows:

$$K_{s,I} = \lim_{\Delta x_{p,b} \to 0} \sum_{\Delta x_{p,b}} \frac{\Delta K_{s,\alpha}}{\Delta K_{s,\alpha} + \Delta K_{s,\beta}}$$

$$K_{s,II} = \lim_{\Delta x_{p,b} \to 0} \frac{1}{\sum_{\Delta x_{p,b}} 1/\Delta K_{s,\alpha}}$$

where $\Delta K_{s,\alpha}$ and $\Delta K_{s,\beta}$ are stiffnesses of series connected infinitesimal prisms of areas $\Delta x_{p,b}$ and $\Delta x_{s,\alpha}$ for the particle and interface member respectively (Fig. 2a), $\Delta K$ is stiffness of a ring of infinitesimal thickness $\Delta h$ (Fig. 2b).

Limits $K_{s,I}$ and $K_{s,II}$ given in Eqn (1) can be expressed as integrals
stiffness surfaces of particles (Fig. 1b), 1.4·10^6 Pa, for the second case state and interface member respectively, where Dz and Db are elasticity constants of the particles and the interface member respectively. These stiffnesses generally depend on a stress–strain state of the materials can be applied to estimate of the stiffness of the connecting element.

3. Some results and brief discussion

The obtained expressions, Eqs (2), (3), and (4), for the stiffnesses Ks,I and Ks,II were verified by comparison with the results of the 3D FEM analysis. Two cases were considered: the first assuming fixed properties of a particle Ep = 40 GPa, and the enveloping values of matrix Ez ∈ [1.4·10^9, 40·10^9] Pa; while for the second one vice versa: Ez = 40 GPa, Ep ∈ [1.4·10^9, 40·10^9] Pa. For both cases radii of particles and interface member are the same and fixed i.e. Rp = Rb = 7.5·10^-3 m and Poison’s ratios also are fixed vp = vb = 0.0. Thus, stiffnesses of materials Dz = Ez and Db = Ep. The distance Lc ∈ [0.1, 0.5, 1.0, 1.5] mm. For the FEM analysis for the first case Ez = 40 GPa and Es ∈ {Ep, 30·10^9, 20·10^9, 10·10^9, 4·10^9, 4·10^8, 4·10^7, 1.4·10^6} Pa, for the second case Ez = 40 GPa and Ep ∈ {Ep, 30·10^9, 20·10^9, 10·10^9, 4·10^9, 4·10^8, 4·10^7, 1.4·10^6} Pa. The stiffness obtained by FEM is denoted hereafter as Ks,FEM. The stiffness Ks,FEM = R / Δl, where Δl is displacement of free plane A (Fig. 3) and R is total reaction force acting at plane B. The FEM analysis was performed by ANSYS 12.

We can see it from Fig. 4 and Fig. 5 when Ep ≥ Ez then Ks,FEM ∈ [Ks,I, Ks,II] and Ks,FEM is closer to Ks,I than Ks,II. When Ez ≥ Ep then for some values of Ep, Ks,FEM ≥ Ks,II ≥ Ks,I.

![Figure 4: The dependence of the stiffnesses on the modulus of elasticity of interface member](image)

Figure 4: The dependence of the stiffnesses on the modulus of elasticity of interface member

![Figure 5: The dependence of the stiffnesses on the modulus of elasticity of particles](image)

Figure 5: The dependence of the stiffnesses on the modulus of elasticity of particles

4. Concluding remarks

Evaluation of upper and lower limits of the stiffness of a normal interaction of spherical particles embedded in a matrix is suggested. Validity of the model was verified and confirmed by the results of the 3D FEM analysis. It was shown that the results obtained by FEM are between the established limits when the matrix is weaker than the particles.

References


MS04

Contact Mechanics

organized by R. Buczkowski, P. Litewka and A. Zmitrowicz
Strain hardening effect on elastic-plastic contact of a rigid sphere against a deformable flat

Łukasz Bąk1, Feliks Stachowicz2, Tomasz Trzepieciński1, Sergei Bosiakov4, Sergei Rogosin5

1,2 Faculty of Mechanical Engineering and Aeronautics, Rzeszow University of Technology
Powsińców Warszawy 12, 35-959 Rzeszów, Poland
e-mail: lbak@prz.edu.pl 1, stafel@prz.edu.pl 2, tomtrez@prz.edu.pl 3
4,5 Faculty of Mechanics and Mathematics, Belarusian State University
4 Nezavisimosti Avenue, 220030 Minsk, Belarus
e-mail: bosiakov@bsu.by 4, Rogosin@bsu.by 5

Abstract

The paper considers the effect of strain hardening on elastic-plastic contact of a rigid ball with an elastic-plastic flat using experiments and finite element software ABAQUS. The strain hardening is an increase in the strength and hardness of the metal due to a mechanical deformation in the microstructure of the metal. Flat tensile samples with different values of strain are considered to study the effect of strain hardening. It was found that strain hardening phenomenon and anisotropy of both friction and material have a great influence on the ball indentation value and the maximal indentation force. The anisotropy of both material and friction conditions influenced the non-uniformity of the stress distribution around the pin axis.

Keywords: contact mechanics, elastic-plastic contact, FEM, micromechanics, strain hardening

1. Introduction

Accurate calculation of contact area is the major problem in the field of tribology and leads to improved understanding of friction and wear. In metal forming operations rough surfaces consist of asperities having different radius and height so it is a difficult task to evaluate the contact area and contact load. The problem is simplified by Hertz [1]. The assumption of surfaces having asperities of spherical shape is adopted to simplify the contact problems.

Based on a model of rigid anisotropic asperities, a theoretical investigation on friction limit surfaces and sliding rules have been carried out by Mróz and Stupkiewicz [2].

Most of friction models are completely defined by the friction conditions which specify a set of admissible contact forces and the sliding rule which stipulates what directions of sliding are allowed. The limit surface is usually assumed to be isotropic predicting a frictional behaviour independent of the sliding direction. For many industrial applications, this assumption seems unrealistic and many experimental studies show that the frictional behaviour can change drastically with the sliding direction, requiring an anisotropic model [3].

The paper considers the effect of strain hardening on elastic-plastic contact of a rigid ball with an elastic-plastic flat using experiments and finite element method. The anisotropic friction model corresponding to experimental results was implemented into a finite element (FE) model built using the ABAQUS software.

2. Material and test method

The experimental study of friction was carried out for DC04 steel sheet metal. The mechanical properties of the sheet metal were determined through uniaxial tensile tests along three directions with respect to the rolling direction (Table 1). The friction properties of the deep drawing quality steel sheets used in the experiments were determined using the pin-on-disc tribometer T01-M. The experimental results of friction tests show that the friction coefficient depends on the measured angle from the rolling direction and corresponds to the surface topography. In the study, the friction coefficient as a function of angular position with respect to the rolling direction of the sheet metal was measured. The anisotropic friction model in FE model was implemented specifying different friction coefficients in two orthogonal directions that coincide with the defined slip directions. Two friction coefficients (μ1 = 0.142 and μ2 = 0.157) were specified, where μ1 is the coefficient of friction in the first slip direction along the rolling direction and μ2 is the coefficient of friction in the perpendicular slip direction.

Continuous indentation tests were performed on DC04 steel sheets with a thickness of 2 mm. The flat samples of 20 mm width and 200 mm length were straightened using uniaxial tensile test to receive different strain levels: 5, 10, 15, 20, 25 and 30%. The indentation tests were performed using modified Zwick Roell Z030 testing machine operated in the compression mode. The applied load versus the crosshead displacement or depth of the indentation were continuously recorded throughout the tests. A maximum load of 60, 80, and 100 N was applied for each sample. A bearing steel indenter with the diameter of 6 mm was used for ball indentation testing.

3. Numerical modeling

Symmetry of the process was assumed in order to reduce the computational time. Only one quarter of blank and the ball with symmetry boundary conditions were modelled. The material of the ball was assumed elastic. An elastic-plastic material model of a sheet was implemented. The elastic behavior is specified in numerical simulations by the value of Young’s modulus, E = 210000 MPa, and of Poisson’s ratio υ = 0.3. In the numerical model, the anisotropy of the sheet material was established. For the blank and hemisphere meshing the 3-dimensional 8-node brick elements were used.

*This work was supported by the European research Agency - FP7-PEOPLE-2013-IRSES Marie Curie Action "International Research Staff Exchange Scheme", grant agreement No. 610547.
Table 1: Mechanical properties of DC04 sheet

<table>
<thead>
<tr>
<th>Sample orientation</th>
<th>Yield stress $R_{p0.2}$, MPa</th>
<th>Ultimate strength $R_m$, MPa</th>
<th>Elongation $A_50$, %</th>
<th>Hardening coefficient C, MPa</th>
<th>Strain hardening exponent n</th>
<th>Lankford’s coefficient $r$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>182.1</td>
<td>322.5</td>
<td>45.8</td>
<td>549.3</td>
<td>0.214</td>
<td>1.751</td>
</tr>
<tr>
<td>45°</td>
<td>197.6</td>
<td>336.2</td>
<td>41.6</td>
<td>564.9</td>
<td>0.205</td>
<td>1.124</td>
</tr>
<tr>
<td>90°</td>
<td>190</td>
<td>320.9</td>
<td>45.6</td>
<td>541.6</td>
<td>0.209</td>
<td>1.846</td>
</tr>
<tr>
<td>Mean value</td>
<td>189.9</td>
<td>326.5</td>
<td>44.3</td>
<td>555.2</td>
<td>0.208</td>
<td>1.461</td>
</tr>
</tbody>
</table>

4. Results

The tensile strain effect on the value of surface roughness parameters is presented in Table 2. The values of the arithmetical mean deviation of the assessed roughness ($Ra$) and ten-point height of the roughness profile ($R_z$) parameters increased with the tensile strain level.

The pin indentation with load equal to 60 N caused a visible effect on the surface profile deformation (Fig. 1). The indentation diameter is equal to ~0.3 mm and indentation depth measured from the highest point of the profile is equal to ~5 µm. The indentation depth value is nearly equal to the $R_z$ parameter value. This fact leads to the conclusion that only the asperities of the sheet surface were deformed.

Table 2: Effect of tensile strain on change of $Ra$ and $R_z$ parameters value of sheet surface.

<table>
<thead>
<tr>
<th>Strain, %</th>
<th>Longitudinal to tension (rolling) direction</th>
<th>Perpendicular to tension (rolling) direction</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$Ra$, µm</td>
<td>$R_z$, µm</td>
</tr>
<tr>
<td>0</td>
<td>0.965</td>
<td>5.043</td>
</tr>
<tr>
<td>5</td>
<td>1.144</td>
<td>5.848</td>
</tr>
<tr>
<td>10</td>
<td>1.182</td>
<td>6.416</td>
</tr>
<tr>
<td>15</td>
<td>1.441</td>
<td>7.913</td>
</tr>
<tr>
<td>20</td>
<td>1.546</td>
<td>8.287</td>
</tr>
<tr>
<td>25</td>
<td>1.583</td>
<td>8.378</td>
</tr>
<tr>
<td>30</td>
<td>1.586</td>
<td>8.584</td>
</tr>
</tbody>
</table>

Figure 1: Optical micrograph of indentation for load of 60 N

As the pin load increases, the plastic zone continues to grow until the edge of the plastic zone reaches the surface near the edge of the contact radius. The maximum value of equivalent plastic strain is found at the subsurface, some distance below the centre of the contact region (Fig. 2). Furthermore, the zone of maximum equivalent plastic strain moves radially from the centre towards the surface inside the maximum contact radius. The anisotropy of both material and friction conditions influenced the non-uniformity of the stress distribution around the pin axis, which was clearly visible for higher values of ball indentations.

Figure 2: Distribution of equivalent plastic strain, indentation depth of 0.013 mm

5. Summary

The tensile strain effect on the change of the value of surface roughness parameters of the steel sheet plate was observed. It caused a different contact area for each strain level of specimens. Under the indentation load of 60 N, only the peaks of asperities on the sheet surface were deformed.

Analysis of the results of simulations of pin indentation shows that as the load increases, the plastic zone continues to grow until the edge of the plastic zone reaches the surface near the edge of the contact radius. It was found that the change of isotropic friction to anisotropic conditions slightly influences the change of equivalent plastic strain distribution.

The presented investigations will be useful in sheet metal forming industry to evaluate the change of deformation-induced surface topography of anisotropic sheets and to select the suitable lubrication conditions.

References

Frictional wear of a wheel-rail system depending on material hardness

Roman Bogacz\textsuperscript{1,2}
\textsuperscript{1} Institute of Vehicles, SiMR, Warsaw University of Technology
Narbutta 84, 02-524 Warsaw, Poland
\textsuperscript{2} IPPT Polish Academy of Sciences
Pawinskiego 5B, 02-106 Warsaw, Poland
e-mail: rbogacz@ippt.gov.pl

Abstract

The question of wheel-rail contact problem in general and wear of wheels and rails are important problems and the view of the phenomena is not synonymous. Majority of railway experts in Europe are convinced that more hard wheel make intensive wear of rails and vice versa, more hard rail makes intensive wear of wheels. Independent of the hardness there are other properties which influence the wear phenomena. The results of experimental investigations made in Railway Institute of Warsaw will be compared with some theoretical results known from literature.

Keywords: wear of rails, wear of railway wheels, contact problems

1. Introduction and literature overview

The wear produced by rolling friction is of particular importance. The wear produced by sliding friction and the wear of solid separated by an intermediate grinding powder are different depending on existence of the lubricant. The true area of contact is relatively very small, that is why the contact stresses are high. It is this feature which gives tribology its special importance and reason for a study. Thus the scientists are concerned with events which are small in size but intense in scale. In order to understand the phenomena, we ought to study the processes of friction and wear.

The development of wear with time is non-linear, i.e. there is a specific period which precedes the wear process. Kragelsky \cite{1} in 1953, developed a classification of wear types based on the consideration of three consecutive stages: interaction of the surfaces on sliding, changes in the surfaces, and surface damage. Since so far there are no clear criteria for wear classification, it is natural that a single classification of wear types has not been so far produced. It is interesting to formulate a proposal of the wear classification based on the nature of the wear particles removed, Peterson \cite{2}. In the case when nominal stress exceeds the fracture stress of a brittle material, particles can be formed by fracture. The advantage of the classification proposed by Archard \cite{3} is that when the mechanism of particle removal is known, the selection of wear resistant materials and wear control techniques is rather simple. The disadvantage of this scheme is that in many situation the wear particle removal process is not known or several processes operate simultaneously. The theory which is used since 1956 is the approach of wear classification proposed by Archard \cite{3}.

The real contact between two solids is discrete and micro-volumes of material are deformed, so that the, the hypothesis of a homogenous isotropic body, widely used in the classical mechanics of deformable bodies, is inapplicable. Holm using the atomic mechanism of wear as his starting point, calculate the volume of substance worn over unit sliding path, i.e.:

\[ W = z(N/HB) \] (1)

where: \( z \) is the probability of removing an atom from the surface when it encounters an atom of the opposing member, \( N \) is load and \( HB \) is Brinell’s hardness of material. Similar dependence was obtained by Archard \cite{3, 4}, on the assumption that the wear particles are hemispheres of radius equal to the radius of a contact point, showed that:

\[ W = (k/3)(N/HB) \] (2)

where \( k \) is the probability of removing a wear particle from the contact point. The value of \( k \) changes over the range \( 10^{-2} \text{–} 10^{-7} \). Burwell and Strang \cite{5} showed that the wear rate is

\[ I_w = K(N/HB) \] (3)

where \( K \) is the wear coefficient which is the ratio of the volume wear rate, expressed in this way, to the true area of contact.

Figure 1: Simplified model of the wheel contact

The motion of wheel driven by a constant moment \( M \) against a resistance is shown as simplified model in Fig. 1. Vertical displacement is parametrically excited by a Hertzian spring in contact. If we assume regularized Coulombian friction and wear according to Archard’s law than the mean value of frictional power \( d\gamma \) (creepage \( s \) multiplied by friction force \( F_T \)), which tends to a limit, multiplied with the wear constant. According to \cite{6}, wear of the rail crown and wheel tread are commonly represented as “mild” and in this wear regime, the wear rate and so-called “wear index” are connected by the equation:

\[ W/Ad = K_s(T \gamma / A) + K_s \] (4)

where \( W \) is the mass of material removed by wear, \( A \) is the contact area between the wheel-rail surfaces, \( d=Vt \), is the distance rolled, where \( V \) is the velocity and \( t \) the elapsed time, \( \gamma \) is the creepage i.e. that wear is proportional to the wear index \( T \gamma / A \), which has the effect of slightly overestimating wear values at small values of wear index, where \( T \) is the creep force.

\[ W/\gamma = K_sTV \] (5)

The creep \( \gamma \) between two bodies in rolling contact is itself the function of the tangential traction. For a circular contact patch,
which seems to be satisfactory approximation, creep is described as follows:

\[ \gamma = 3\mu N(4 - 3\nu)(1 - (1 - T / \mu N)^{1/3})/16G a^2 \]  
(5)

where \( \mu \) is the coefficient of friction, \( N \) is the normal load, \( a \) is the radius of the contact patch and \( G \) is the shear modulus. This equation is relevant only to rolling contact i.e. the tangential traction \( T < \mu N \). If the traction was greater than this, there would be full slip between wheel and rail. The work carried out by McEven and Harvey in [7], focused also on the theory that material loss is proportional to the energy dissipated in the contact zone, i.e. to the product of creep force and creepage. The relationship varies depending on whether the wear is in the mild or severe wear regime or in the transition between these regimes. The material loss relationship for a wheel/rail is as follows:

\[ \gamma = \frac{25\gamma }{D} \]  
(6)

\[ \gamma > 200 N \]  
(material loss \( (1.19\gamma - 154)/D \))

where the loss is expressed in mm\(^2\) loss of area from any radial section through the profile per kilometer rolled and \( D \) is the diameter of the wheel in millimeters.

The majority of investigations (except [9] and some fundamental, like [1,2,3]) are not devoted to the relation of wheels and rails wear rate as dependent on material hardness [4,5,6,7]. This induces impression that the wear behaviour the rail does not depend on hardness of wheels. The experimental investigations described in the next part of paper show an essential dependence on material hardness, more ever the dependence is opposite as it is commonly supposed in Europe.

2. Experimental investigations of wheel/rail wear

Majority of railway experts are convinced that more hard wheel make intensive wear of rails and vice versa, more hard rail make intensive wear of wheels. The proportionality between wear and frictional work is used in various wear models. It is hard to compare the experimental results and the constants which were used in the simulation calculations because they were carried out under different assumptions and conditions. The wear parameter used in theoretical study [2,3,4,5] for the steel/steel frictional pair, depends on the shear modulus \( G \) and on the material hardness. In the case of steel/steel pair, with of given frictional parameter \( \mu \), the wear is inverse proportional to the hardness of individual part (wheel/rail).

The wear testing was conducted on the EMS 60 test stand at Railway Institute (IK). Five types of new, 920 mm diameter wheels and two types of new 60E1 rails (of hardness 281 and 284 HBN) were used for each wear test, and each wheel type was tested three times. For testing the forged/rolled wheels manufactured by a leading European manufacturer, having a minimum hardness of 235 BHN and three kinds of pressure poured cast wheels of minimal hardness.

The experimental investigations described in the next part of paper show an essential dependence on material hardness, more ever the dependence is opposite as it is commonly supposed in Europe.

<table>
<thead>
<tr>
<th>wheel type</th>
<th>average wear of rails [mm]</th>
<th>wear of wheels [g]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Forged/rolled UIC ER7</td>
<td>3.86</td>
<td>898</td>
</tr>
<tr>
<td>Cast ER7</td>
<td>1.46</td>
<td>350</td>
</tr>
<tr>
<td>Cast Class B</td>
<td>1.42</td>
<td>334</td>
</tr>
<tr>
<td>Cast Class C</td>
<td>1.26</td>
<td>292</td>
</tr>
</tbody>
</table>

3. Conclusions

Wheel/rail wear testing shows that the Class C pressure poured cast wheel of maximal hardness is not more likely to damage rails and infrastructure than relatively soft currently used forged/rolled wheels. Wheel wear and rail wear are significantly less for the more hard cast ER7 and cast Class B wheels, than for more soft ER7 forged/rolled wheels.

References

Investigations on the chip shape and its upsetting and shortening ratios and surface roughness in partial symmetric face milling process of aluminium alloy AW-7075 and the simulation of the process with the use of FEM

Jaroslaw Chodor1, Lukasz Żurawski2
1, 2Faculty of Mechanical, Koszalin University of Technology
Raclawicka 15-17, 75-620 Koszalin, Poland
e-mail: jaroslaw.chodor@tu.koszalin.pl1, lukasz.zurawski@tu.koszalin.pl2

Abstract

The paper presents the results of experimental research of milling aluminium alloy AW-7075 in low temperature. Surface roughness, chips shape and their ratios of upsetting and shortening after process are shown. The investigations were carried out using the Sandvik CoroMill R245-125Q40-12M milling head and exchangeable R245-12T3M-PM4030 carbide plates with superfinishing surface. Numerical analysis of milling (cutting) process were conducted in ANSYS/LS-PrePost programme. Technological and geometrical parameters of the tool and the workpiece material were identical to these in the experimental tests. Chip shapes and the ratios of upsetting that were obtained experimentally, then numerically were compared.

Keywords: chip, ratios of upsetting and shortening, milling, cutting, simulation, numerical analysis, ANSYS/LS-PrePost, FEM

1. Introduction

One of the problems of modern technologies is the fulfillment of growing requirements connected with the operation of plant and machinery, that concern an increase of their durability and reliability. This is determined by an appropriate formation of the top layer. Its condition is essential because almost all tribological and wear processes occur on the object surface. The properties of the top layer after milling depend chiefly from the variant and realization conditions of the process.

In order to analyse in a comprehensive manner the milling process, it is necessary to develop an adequate mathematical model and numerical methods in relation to its solution. In connection with this, the present paper makes use of the physical and mathematical models of the milling process which were developed in the research team. The algorithms of the solutions of the discrete systems obtained of equations including appropriate initial and boundary conditions were developed on their basis. The milling process was considered a geometrically and physically non-linear boundary and initial problem with an assumption of the occurrence of non-linear moving and boundary conditions that are variable in time and space, whereas these conditions are unknown in the contact area between the head and the workpiece material. The algorithms that were prepared in this way were implemented in the ANSYS software.

The main purpose of the study was to develop an application in the ANSYS system to allow one to observe those phenomena that occur in extremely small areas and that take place with high velocities, which last very short time, and yet they determine the results of the milling process. Similar problems occurs in other works [1,2,3,4,6]. These phenomena include among others the chip formation phenomenon, which in the conditions of an experiment are hard to observe due to the dynamics of the process. A numerical analysis allows for a detailed observation of the chip formation phenomenon and also an observation of the distribution of the intensity of stresses, the intensity of strains and distribution of temperature in any time and at any place of the process [5,7].

The research problem of the study is the determination of the chip shape as well as the determination of the upsetting ratio \( \lambda_h \) and chip shortening ratio \( \lambda_l \) in the process of cylindrical, face, symmetrical and incomplete milling in low temperature. The shapes of chips and the values of upsetting ratios from the experimental tests were compared with the chip shapes and the values of upsetting ratio obtained in computer simulations. In computer simulations, the technological parameters, the tool geometry and the parameters of the workpiece material were identical with those from the experimental tests. On the background of a comparison of chips, the correctness was to be found of the applications developed and of the preparation of numerical analyses.

2. Experimental researches

The investigations related to the chip shape and its ratios of upsetting and shortening were carried out with the use of the following devices and tools [8, 9]: Sandvik CoroMill R245-125Q40-12M milling head, diameter Dc=125 mm with the number of blades \( z=8 \).

Specimens from aluminium alloy AW-7075 has size 100×173 [mm]. In the experiment, the versatile FWD32U vertical milling machine was used.

Figure 1: Exemplary form of chips received during experimental researches of milling Aluminum alloy AW-7075

Partial symmetric face milling process in low temperature (-180°C) was a discontinuous process. Milling head had a machined thin layer of material, then returned to initial position and later machined next layer of material (Fig. 2).
Blades works is characterized in such a way that contact every of them with machined surface depends from contact angle $\Psi$ which is between points: $0, S_p$ – initial machining and $0, S_k$ – final machining. Value of this angle is changing in the range of $0^\circ \leq \Psi \leq 180^\circ$ and depends from width of machined material $B$ and diameter of tool $D_c$ (Fig. 2).

After every pass of a milling head chips were collected. They were assigned to length of cut path $L_s$. Form of chip, upsetting ratio, chip shortening ratio and surface roughness were determined.

3. Numerical analysis

Milling is considered as a geometrical and physical nonlinear initial and boundary problem. The analytical solution of this problem like: determination of states of deformations and stresses in the any moment of duration of the process is impossible. Therefore this problem by finite element method (FEM) was solved. Also tried solved this problem using SPH method (Smoothed Particle Hydrodynamics).

Chip shapes in numerical simulations are similar with experimental chips received after milling process. Upsetting ratio, chip shortening ratio also were on the similar level. That confirms the justifiability of the use of computer simulations and their reliability.

References

Validation of a contact layer element for soil-structure interactions within Abaqus software environment

Pawel Dziewiecki1,2, Christian Weißenfels2, Peter Wriggers3

1,2,3Gottfried Wilhelm Leibniz Universität Hannover, Institute of Continuum Mechanics
Appelstraße 11, 30167 Hannover, Germany
e-mail: dziewiecki@ikm.uni-hannover.de 1, weissenfels@ikm.uni-hannover.de 2, wriggers@ikm.uni-hannover.de 3

Abstract

Keywords: soil-structure interaction, contact mechanics, hypoplastic constitutive law, continuum-shell formulation

1. Introduction

The main goal of the project is the realistic simulation of installation processes in soils using numerical simulation tools. The boundary conditions, for one are the contact forces between soil and structure. For correct prediction of these processes loads a suitable friction model is required. During the relative movement of a structure contact with a rough surface on soil a shear zone actually evolves within the sand, which is located directly at contact surfaces. Thus the interaction behaviour between sand and the structure results in varying coefficient of friction, which is assumed to be a quantity dependent on the stress state within the soil near to the contact surface. This assumption leads to an extension of the classical formulation of friction laws used within the contact mechanics framework related to the inelastic material behaviour. Within the proposed approach the thin layer is modelled using the contact-shell framework proposed in [1].

2. Models

Following this idea a layer with an intrinsic height h consisting of continuum-shell contact elements, depicted in Fig. 1, is introduced. This introduction requires an adaptation of the classical weak form in the case of contact:

\[
G(u, \eta) = \left( \int_{\varphi(B)} (\delta \varepsilon \cdot \sigma - \eta \cdot \rho_0 b) dv + \int_{\varphi(\partial B^*)} \eta \cdot t da \right) + \int_{\varphi(\partial B^*)} (t_n \delta g_n + t_t \cdot \delta g_t) da ,
\]

where \( u \) and \( \eta \) are displacements and virtual displacements, respectively, \( \varepsilon \) strains, \( \sigma \) stresses, \( \rho_0 \) density, \( t \) vector of external forces, \( t_n \) and \( t_t \) contact stresses and \( g_n \) and \( g_t \) normal and tangential contact gap, respectively. The symbols \( \varphi(B) \) and \( \varphi(\partial B^*) \) describe the domain, the Neumann boundary and the contact boundary, respectively. The last term in eq. (1) is the contact contribution enforcing the constraints of the non-penetration and the stick-slip condition, respectively. For the introduced continuum-shell contact element this term results in an equivalent formulation using the complete three-dimensional stress-strain state:

\[
\int_{\varphi(B)} \delta \varepsilon \cdot \sigma dv = \lim_{h \to 0} \int_{\gamma} \int_0^h \delta \varepsilon d \xi \cdot \sigma da = \int_{\varphi(\partial B^*)} (t_n \delta g_n + t_t \cdot \delta g_t) da ,
\]

with the Almansi strain \( \varepsilon \), intrinsic height \( h \) and contact area \( \gamma \). For modelling the soil behavior a hypoplastic law developed in [2], extended and reformulated in terms of classical return mapping algorithms in [3] (in particular for stress dependent inelasticity), is used. Hypoplastic material laws represent the volume changing effects of sand due to loading very well, which is specific and crucial for the modelling of granular media. The hypoplastic law has the following form

\[
\sigma = L \cdot (\dot{\varepsilon} - Y n\|\dot{\varepsilon}\|),
\]

where \( L = \lambda_e p L \) is elasticity tensor, which depends on the hydrostatic pressure \( p \), with Poisson’s ratio \( \nu = 0 \), \( L \) a 4th order unit tensor, \( \lambda_e \) the barotropy parameter of the model and \( n \) the normal projection tensor on the yield surface. The quantity \( Y \) is an interpolation of the degree of non-linearity between elasticity and plasticity. Eq. 3 contains in a compact form the ingredients for describing of inelastic material behavior. Discretized forms of the equations result in contributions which are implemented into Abaqus via user subroutines UINTER and UMAT.

3. Validation

The proposed approach is validated on a pull out of a wall testing device documented in [4] and generally sketched in Fig. 2. Within this experiment the evolving of normal and tangential contact force, and thus the coefficient of friction as a ratio of both, is investigated. Fig. 3 shows results for simulation of this test using proposed model. At this point a simplified material model is used as a first approach. The model is evaluated on one side for a smooth wall and on the other side with one of the wall segments having increased roughness (for set up see Fig. 2). Since for the wall with all segments of the same properties contact stresses are smooth and for the wall with one rough segment contact stresses, particularly the tangential stress in the movement direction, are higher, the qualitative results are feasible.

*Corresponding author.
4. Application

The proposed model is developed for applications to geotechnical processes like pile installation or sheet pile wall installations in areas dense covered by buildings and the mechanical impact of this kind of processes to the nearest surroundings. At the present stage the work is dedicated to a realistic simulation of a former large construction site applying technology of installation in soils.

5. Conclusions

The simulation model presented in this contribution is validated on existing large scale testing device. However since soils are naturally containing water, further development should include partial saturation. This is another important point connected to the modelling of soil-structure interaction because of the possible occurrence of liquefaction. As a consequence of cyclic loading to the soil caused by installing of moving parts of construction gradually loss of load carrying capacity may occur. This loss of capacity is caused by progression of water saturation of the nearest environment of the installation. This saturation can also amplify the above mentioned volumetric effects. Because of this dependencies the constitutive model has to be extended by introducing an additional independent variable. Following the concept proposed in [5], the matric suction \( s = u_a - u_w \), defined as the negative water pressure \( u_w \) measured in excess of air pressure \( u_a \), will be used here.

![Figure 1: Solid shell element with intrinsic coordinate system used for defining the kinematics of the contact area](image1)

![Figure 2: Pull out test geometry](image2)

![Figure 3: Results for a simplified material model (Drucker-Prager material) after wall moved upwards. Upper line: complete wall smooth, lower line: with one rough wall segment. In both lines, from left to right, contact stresses \( t_{11}, t_{22}, t_{12} \). ( Fragments of sand block removed in postprocessing for a better visibility of the wall )](image3)

References


Modeling of adhesive contacts in composite materials

Piotr Fiborek
Instytut of Fluid-Flow Machineries, Polish Academy of Sciences,
Street Fiszera 14, 80-231 Gdańsk, Poland
e-mail: pfiborek@imp.gda.pl

Abstract

The paper presents structural defects in adhesive bonds of composite materials of a significant influence on the strength of these materials. Vibration-based method detecting these defects is described in the paper. However, there are still no adequate models of damages in adhesive contacts. It is the main subject of our investigation.

Keywords: damage detection, composite materials, adhesive contact

1. Introduction

The growing demand for lightweight and durable structures in aircraft and aerospace industry affected the intensive development of composites and nowadays they are used in almost every field of technology: from building materials to the medical implants. However, the lack of valuable non-destructive methods for the quality assessment of adhesive bonded composite components strongly limits the further use of composites, especially in parts with high safety requirements. Therefore, many research centers are involved in numerous scientific studies to develop effective methods of quality control for these materials.

2. Adhesion models

Adhesion is a phenomenon that occurs when surfaces of two bodies come sufficiently close in contact. It can be estimated with the aid of a work required to separate those bodies.

A number of models of adhesion were developed over recent years among which the most popular are: (a) Johnson–Kendall–Roberts model (JKR, 1964–1971), (b) Derjaguin, Muller, Toropov model (DMT, 1975) or (c) Frémond model (1989).

Figure 1: Illustration of adhesive contact: (left) initially applied, normal, compressive force, (right) normal tensile force separating surfaces from contact.

JKR and DMT models consider contact of elastic spheres and based on Hertz theory estimate adhesive energy. The JKR model applies to contact between two spheres with large curvature radius and small stiffness while the DMT model is applied to contact between two spheres with small curvature radius and high stiffness. According to JKR [6] and DMT models [1] contact radius is given by

$$a^3 = \begin{cases} \frac{R}{K}(F + 3\gamma \pi R + \sqrt{6\gamma \pi RF + (3\gamma \pi R)^2}) & \text{acc. to JKR} \\ \frac{R}{K}(F + 2\gamma \pi R) & \text{acc. to DMT} \end{cases}$$

and pull-off force is given by:

$$F_{\text{pull-off}} = \begin{cases} -\frac{3}{2}\gamma \pi R & \text{acc. to JKR} \\ -\frac{4}{5}\gamma \pi R & \text{acc. to DMT} \end{cases}$$

where, $F$ is applied force, $\gamma$ is surface energy, $R = R_1 R_2 / (R_1 + R_2)$ and $K = 4/3\pi (k_1 + k_2)$ where $R_1, R_2$ are radii of curvature and $k_1, k_2$ are the elastic constants of the material of each sphere.

Different approach to the subject was presented by Frémond introducing a variable to describe the state of contact: the ratio $\beta$ of active bonds called the intensity of adhesion [4]. The value of $\beta$ is $0 \leq \beta \leq 1$ while $\beta = 0$ means no adhesion (all the bonds are broken), and $\beta = 1$ means total adhesion (all bonds are active).

3. Structural defects of adhesive bonds in composite materials

Strength of adhesive bonds is usually the weakest link in composite material or in the entire engineering constructions composed by many components. Moreover, defects which arise as a result of mistakes during the manufacturing process or under storage and in-service operation have a significant effect on reducing the joint strength.

Defects in adhesive joints can be divided into two basic groups: (a) defects inside the adhesive layer, (b) defects in the interface between adherend and adhesive.

3.1. Defects inside the adhesive layer

Porosity and voids are the mainly occurring defects in the adhesive layer as a result of insufficient application of adhesive by entrapping air or volatile substances such as water vapour. The pores and voids decrease the strength of the bonds and provide a place where moisture can be accumulated.
Another disadvantage of an adhesive layer are cracks that occur by inadequate curing conditions or under the influence of external forces in the case of brittle joints [8].

3.2. Defects in the interface between adherend and adhesive

Delamination, debonding and weak bonds (also called “kissing bonds”) are the most common defects occurring in the interface between adhesive and adherend. Delamination is a defect of the composite structure, which leads to separation of the layers of reinforcement or plies due to poor processing during production, impact in service, or some other means. In contrast to delamination, debonding is defined as a separation of the composite material from another material. Whereas, “kissing bonds” is defined as a perfect contact two surfaces where there is no shear stress between them [5].

In addition to these cases, the quality of the composites structures can be decreased by contamination of the substrate. For example, the silicon which is remains of mould release agent or Skydro hydraulic fluid used in hydraulic systems of aircraft navigation can occur on surface and react with water to form phosphoric acid that can etch material [2].

3.3. Models of damages in adhesive contacts

Different types of defects require different approach to modeling of these defects. All kinds of failures such as delamination, debonding, cracks and voids can be modeled as a material discontinuity, whereas interface elements can be utilized to model all types of inclusions. Moreover, model of damages in composite material can be also realized by means of a representative volume elements (RVE).

4. Vibration-based methods for damage detection

One of the most popular defect detection technique is a vibration-based method. It usually applies computational, dynamic models of structures and compares them with experimental data to determine damage location. Various response characteristics of the structure such as natural vibrations or forced vibration can be analyzed with the aid of dynamic models [9].

4.1. Natural vibrations

The defects in structure affect the modal parameters which depend on physical properties of the structure such as stiffness and mass. The change in stiffness produces a characteristic decrease of natural frequencies and change of the natural mode shapes. Modal parameters can be easily and cheaply obtained by some form of transducer which monitors the structural response to artificially induced excitation forces or ambient forces in the service environment [7]. There are two principal disadvantages of this method: (a) the influence of small damages on modal quantities is rather low and can be masked by experimental uncertainties and (b) errors in the identification procedure by the assumption of linearity of structure can occur.

4.2. Impedance domain

Damage in the structure causes changes in the structural impedance of the electro-mechanical system. Piezoelectric transducers are mounted on the structure in order to excite and detect elastic waves. The impedance method is a very sensitive method working in the higher frequency range.

4.3. Forced vibrations

An alternative technique to the previous methods is a method using time histories of the input and vibration responses of the structure. A general idea of this method is that damages increase an amplitude of excited vibrations. This method is not limited by the assumption of linearity in contrast to modal analysis but it may required considerable computational effort for the calculation.

5. Conclusion

Adhesive bonding is the most optimal technique of joining composite materials but there is still a lack of adequate quality assurance procedures. Therefore many research centers focus on developing advanced methods of quality assessment e.g. team with Prof. W. Ostachowicz as a leader from Institute of Fluid-Flow Machinery, Polish Academy of Sciences participates in the international research project ComBoNDT. Modeling with regard to adhesive contact can improve better determination of positions, shapes and dimensions of a single or a group of defects in adhesive bonds. This is the subject of our investigations.

References

Analysis of the influence of differences in strength parameters of steel S235 on passive safety of lighting columns

Tomasz Ireneusz Jedliński¹, Jacek Buśkiewicz²

¹ Research and Development Specialist, Europoles sp. z o. o.
Kasztelanska 39, 62-571 Kragola, Poland
e-mail: jedlinski.tomasz@gmail.com

² Faculty of Mechanical Engineering and Management, Poznań University of Technology
Piotrowo 3, 60-965 Poznań, Poland
e-mail: jedlinski.tomasz@gmail.com, jacek.buskiewicz@put.poznan.pl

Abstract

Available designs of lighting columns cause different effects on road safety. A properly designed and manufactured column can reduce the range of injuries sustained in a road accident, and may even save human life. The article describes the impact of strength parameters of steel used in the production of the whole construction on passive safety. The material S235 has partially defined strength boundaries, however, due to their large extent one can get much harder material. In this case, the developed products have different functional properties from the expected ones.

Keywords: passive safety, crash test, EN 12767, steel strength

1. Introduction

Nowadays the main objective in the design of new roads and their surroundings is to ensure maximum safety for the users. Such approach is reflected in the guidelines of the design of modern lighting columns are described in European standard EN 12767 "Passive safety of support structures for road equipment. Requirements, classification and test methods".

Literature review on the general issues of strength of materials was related mainly to the books [6]. However, the research part was based on scientific studies and articles in specialized journals [1,2,3,4,5] due to the relatively narrow industry.

2. Equipment – measuring apparatus

The equipment required differs in two parts of the project:

- In the first draft, the column was analyzed using static software of Europoles Company. The software allows to calculate the static load of columns having a constant force derived from suspended loads and a force exerted by wind pressure.
- Produced columns were then tested on two test posts:
  - Mechanical properties of steel are determined using testing machine. During the test, the extension of samples was measured by the extensometer, and the strength with the strain gauge force sensor placed on the hydraulic cylinder.
  - The position of static and dynamic tests of the complete lighting columns enables mounting the columns to a rigid base in the horizontal position and loading them anywhere with the use of a hydraulic cylinder. The actuator is controlled automatically and the preset load program is carried out by the driver. During the test, deflection and bending moment of the column are measured.
  - Crash track for static safety testing consists of a specially designed truck imitating the behavior of a conventional vehicle on which the measuring apparatus is installed. During the test overloads are measured in three axes in the range of ± 600g and the rotational velocity of the vehicle up to 50 rad/s. All data is stored in a computer with a sampling rate of 10000 Hz. In addition, an area of 6 m from and 12 m behind the point of the impact is recorded by two high-speed cameras (500 frames/sec and a resolution of 800 x 600 pixels). All data recording is triggered by a contact sensor placed on the surface of the column.

3. Crash test - measurement parameters

All tests were conducted in accordance with the assumptions of the standard EN 12767 "Passive safety of support structures for road equipment. Requirements, classification and test methods". Columns are divided with respect to the absorption of energy of the vehicle (Fig. 1 and Table 1).

The driver's safety is defined within each category from 1 (lowest safety level) to 4 (highest safety level) (Table 2) depending on Acceleration Severity Index (ASI) and the Theoretical Head Impact Velocity (THIV) [2,3]. Both indicators are calculated in accordance with EN 12767. ASI depends on the g-force accelerations in all axes, while THIV is the theoretical speed of the driver's head when hitting the steering wheel, and is calculated from the negative g-forces and vehicle speed [4].

Figure 1: Non-energy-absorbing (NE), low energy-absorbing (LE) and high energy-absorbing (HE) lighting columns [3]
4. Lighting pole - description and testing

The tested lighting column was sunk into a concrete foundation 0.5 m deep and attached to it by a clamp. The column had a height of 4.5 m, a peak diameter of 76 mm, the convergence of 14 mm/m. The column was produced of S235 steel 3 mm thick. A lighting lamp of a weight 12 kg was mounted at the top of the column. The total weight of the column had a height of 4.5 m, a peak diameter of 76 mm, the convergence of 14 mm/m. The column was produced of S235 steel 3 mm thick. A lighting lamp of a weight 12 kg was mounted at the top of the column. The total weight of the column is 47 kg.

The columns were made of steel of the same grade having different strength properties. The first columns were made of S235 steel 3 mm thick. A lighting lamp of a weight 12 kg was mounted at the top of the column. The total weight of the lighting columns was 47 kg.

The columns were made of steel of the same grade having different strength properties. The first columns were made of S235 steel 3 mm thick. A lighting lamp of a weight 12 kg was mounted at the top of the column. The total weight of the lighting columns was 47 kg. The column made of the material with lower strength parameters is assigned to category 70NE2 whereas the other column is assigned to 70LE3.

4. Lighting pole - description and testing

The tested lighting column was sunk into a concrete foundation 0.5 m deep and attached to it by a clamp. The column had a height of 4.5 m, a peak diameter of 76 mm, the convergence of 14 mm/m. The column was produced of S235 steel 3 mm thick. A lighting lamp of a weight 12 kg was mounted at the top of the column. The total weight of the column is 47 kg.

The columns were made of steel of the same grade having different strength properties. The first columns were made of S235 steel 3 mm thick. A lighting lamp of a weight 12 kg was mounted at the top of the column. The total weight of the lighting columns was 47 kg. The column made of the material with lower strength parameters is assigned to category 70NE2 whereas the other column is assigned to 70LE3.

Figure 2: Pictures 100, 150 and 200 m/s after the crash

5. Conclusions

The differences in the crash test results were so significant that the columns have been assigned to different categories of energy absorption and levels of driver safety. A wide range of strength parameters of steel S235 and the lack of restrictions as to their controls before crash testing undermines their credibility and meaning of the performance.

References

Numerical analysis of contact between 3-D beams with deformable circular cross sections

Olga Kawa¹, Przemysław Litewka²

¹² Institute of Structural Engineering, Poznan University of Technology
Piotrowo 5, 60-965 Poznan, Poland
e-mail: olga.kawa@put.poznan.pl, przemyslaw.litewka@put.poznan.pl

Abstract

In the paper frictionless contact between three-dimensional elastic beams with deformations at the contact zone is analysed. It is assumed that the beams analysed undergo large displacement, the strains remain small and the beams cross-section are deformed. The deformation of the cross-sections follows from Hertzian solution. The appropriate kinematic variables are defined and discretised using the finite element methodology. To enforce the normal contact constraints the penalty method is applied.

Keywords: contact, beams, finite element method, consistent linearization, deformed cross-sections

1. Introduction

The main purpose of computational contact mechanics is to provide numerical tool to properly describe the physical behaviour of bodies coming in contact, and especially the deformation and forces acting in the vicinity of the contact interface. There are several contributions which are related to beam-to-beam contact, e.g. [3, 4, 5, 7, 8].

To include the cross-section deformation the classical analytical result from Hertzian contact between two elastic cylinders representing the contacting beams has been proposed in [3], [4]. In the paper the authors show a different way to include such cross-section deformations at the contact zone. In the new element the deformation is introduced replacing the penalty parameter with the contact stiffness.

2. Kinematic relations

We consider two beams with circular cross-sections coming into contact. To detect contact we have to define the penetration function for two beams which for beams with circular cross-sections can be written as

\[ g_n = d_n - r_m - r_s , \]  

where \( d_n \) is the minimum distance between the beams, \( r_m, r_s \) – the radii of the beams cross-sections.

The function of the penetration can be used as the criterion of contact. The contact condition can be defined as

\[ g_n = d_n - r_m - r_s \leq 0 . \]  

To evaluate (2) we have to find two closest points lying on two curves representing the beams axes (\( m \) and \( s \)). After finding the position vectors \( x_{mn} \) and \( x_{ns} \) of the points \( C_{mn} \) and \( C_{sn} \) one can calculate the minimum distance between the beams \( d_n \) and define the penetration function.

2.1. Contact points

Location of the points on the curves in the 3D space is defined by local curvilinear co-ordinates: \( \xi_n \) for first beam and \( \xi_s \) for the second one (Figure 1). The current configuration for each point on the curve is expressed as

\[ x_m = X_m + u_m , \]
\[ x_s = X_s + u_s , \]

where: \( X_m, X_s \) are the position vectors for points at the initial configuration, \( u_m, u_s \) are the displacement vectors. For the closest points \( C_{mn} \) and \( C_{sn} \) the position vectors \( x_{mn} \) and \( x_{ns} \) must fulfil simultaneously the orthogonality conditions

\[ (x_{mn} - x_m) \cdot x_{mn, m} = 0 , \]
\[ (x_{ns} - x_s) \cdot x_{ns, s} = 0 , \]

where:

\[ x_{mn, m} = \frac{\partial x_{mn}}{\partial \xi_m} , x_{ns, s} = \frac{\partial x_{ns}}{\partial \xi_s} . \]

The solution, i.e. the pair of co-ordinates \( \xi_{mn} \) and \( \xi_{sn} \), which describe the location of the points \( C_{mn} \) and \( C_{sn} \), is obtained using the Newton method. Linearization of the conditions given in Equations (4) leads to a set of two equations, which allow for calculation of co-ordinate increments \( \Delta \xi_{mn} \) and \( \Delta \xi_{sn} \) [7].

After finding the position vectors \( x_{mn} \) and \( x_{ns} \) of the points \( C_{mn} \) and \( C_{sn} \) may be computed the minimum distance between the beams

\[ d_n = \| x_{mn} - x_{ns} \|. \]
\[ F_N = \frac{4}{3} E \cdot R^{1/2} \cdot d^{3/2} \]  

where: \( R \) is the value of the effective radius, \( E \) – the mean Young’s modulus, \( d \) – the change of the radii \( r_m, r_s \) due to the deformation of the cross-section.

The equations are given for the circular contact region so we have to do some approximation. In Equation (7) we have to use the value of the effective radius

\[ R = \sqrt{r_s \cdot r_m} . \]  

If the bodies are elastic, then the following expression for the mean Young’s modulus \( E \) must be used

\[ \frac{1}{E} = \frac{1-v_s^2}{E_s} + \frac{1-v_m^2}{E_m}. \]  

The value of \( d \) present in Equation (6) is at the same time the penetration function (Figure 2). The relation between the penetration which is defined by gap function (1) and the value \( d \) can be written as

\[ g_N = -d, \]  

where

\[ d = \frac{9}{16} \frac{F_S^{2/3}}{E^{1/3} \cdot R^{1/3}}. \]  

Thus, the minimum distance between the beams is

\[ d_a = r_m + r_s - d. \]  

In the iterative solution procedure the radii change \( d \) in the current step can be evaluated using the normal force and the normal gap \( g_{Np} \) from the previous step.

2.3. Kinematic variables for contact points

The solution of a frictional contact problem for two bodies involves finding a minimum of the potential energy functional

\[ \min \Pi = \min (\Pi_1 + \Pi_2 + \Pi_c). \]  

The contact contribution takes the form

\[ \Delta \Pi_c = F_S \Delta \delta_N. \]  

Substituting from (6) into (12) we obtain

\[ \Delta \Pi_c = \frac{4}{3} E \cdot R^{1/2} \cdot g_N^{3/2} \Delta \delta_N. \]  

The related linearisation required for the Newton solution scheme for the non-linear contact problem reads

\[ \Delta \delta N = \left( \frac{4}{3} E \cdot R^{1/2} \cdot g_N^{3/2} \right) \Delta \delta_N \]  

\[ + \left( \frac{4}{3} E \cdot R^{1/2} \cdot g_N^{3/2} \right) \Delta \delta_N. \]  

The variation, linearisation and second variation of the penetration function \( \delta_N, g_N, \Delta \delta_N \) are computed in the same way as in the analysis without the cross-section deformation [5]. The finite element discretisation follows the same lines as presented therein, too.

3. Concluding remarks

The deformation of the cross-section represented by the distance \( d \) is included in contact formulation. In the new elements the deformation is introduced by replacing the penalty parameter with the contact stiffness. That leads to obtain a different formulation of gap function than the one presented in [5]. Numerical examples and comparison of the results from this new and earlier approach [5] will be presented at the conference.

References

Electro-mechanical multiple-point beam-to-beam contact

Przemysław Litewka*
Institute of Structural Engineering, Poznan University of Technology
ul. Piotrowo 5, 60-965 Poznań, Poland
e-mail: przemyslaw.litewka@gmail.com

Abstract

The electro-mechanical contact is analysed for beam-like electric conductors getting in touch at acute angles. The concept of multiple-point contact is used to better cover the situation when contact cannot be considered point-wise. The electric contact is introduced with the constraints yielding from the Ohm's law and the phenomenon of long constriction present in the situation when the electric current flows through a contacting spot with dimension much smaller than the regular dimensions of the contacting conductors. The governing equations are written separately for the electric and mechanical fields using voltages and displacements as principal unknowns, respectively. The consistent linearisation leads to the tangent stiffness matrix for the problem which features semi-coupling with the electric field depending on the mechanical one.

Keywords: beam-to-beam contact, multiple-point approach, electro-mechanical coupling, Ohm's law

1. Introduction

Contact between beams is a special case in the 3D contact analysis. This topic was initiated by Wriggers and Zavarise in [9] and continued in [4,5,10] where contact without and with Coulomb friction for beams of circular and rectangular cross-sections was covered. Further development concerned inclusion of thermal and electric coupling, see [1]. A rigorous approach to the question of point-wise contact was also suggested by Konyukhov and Schweizerhof in [3]. The authors focused on the closest-point projection procedure, which for the beam-to-beam contact leads to the orthogonality conditions.

One of the key assumptions in the majority of those beam-to-beam analyses was, that the closest points of two curves can always be uniquely determined, at least locally. In other words, the point-wise contact between beams was assumed. However, such an approach may fail in some cases. Hence, a general approach must include possibilities when the contacting beam-like objects form acute angles and are parallel or conforming, see Fig. 1. The issues related to the existence and uniqueness of the closest points location were discussed in detail in [3].

The beam-to-beam contact finite element presented in this paper is the further enhancement of the multiple-point contact formulation [6,7] to include the electro-mechanical coupling. Section 2 includes a brief description how the additional contact pairs are defined. In Section 3 the principle variables for electro-mechanical contact are introduced. Section 4 brings equations of the electro-mechanical contact and their linearisation is presented, too. Then the finite element discretisa-

Figure 1: Non point-wise contact between beams: a) beams with axes crossing at acute angles, b) conforming beams

2. Additional contact pairs

Modelling of the beam-to-beam contact for almost conforming beams has been a topic of the papers [6,7]. The present formulation for the electric-mechanical interaction follows that pattern. Two additional contact points denoted by the subscripts b and f defined by the local co-ordinates

\[
\bar{\varepsilon}_{sb} = \bar{\varepsilon}_{rf} = \frac{\varepsilon_{s} - \varepsilon_{f}}{l_{c} \sin \phi}
\]

are found on the beam marked by s using the local co-ordinate shift coming from the numerical experiments [8]

with respect to the central points initially found using the orthogonality conditions [9]. In (2) \( l_{c} \) is the length of the beam element within which the contact point lies, \( r_{m} \) is the radius of the beam circular cross-section and \( \phi \) is the angle between the beam axes at the centre point.

Then the unilateral orthogonality conditions on the beam \( m \)

\[
(x_{sb} - x_{sb}) \cdot x_{sb,me} = 0
\]

are used to find the conjugate points completing the definition of three contact pairs. Vectors \( x \) in (3) and (4) represent the position vectors of additional contact points on the beams \( m \) and \( s \), see [6].

3. Electro-mechanical variables for contact

The mechanical variables for the displacement-based FEM formulation of the problem in the theory of elasticity are displacements, they are used to define the mechanical contact variable – the normal gap \( g_{s} \). This definition for three contact pairs is presented in [6].

The electric variable used in [1] for a point-wise formulation of the electric contact is the electric voltage. Thus

*The research reported in this paper was partly financed by the internal university grant 01/11/DSPB/0300.
electric contact variables may be defined – the voltage gaps, for the additional contact pairs:

\[
g_{N} = V_{N_{b}} - V_{N_{h}}  \\
g_{F} = V_{F_{y}} - V_{F_{y}}
\]

Using the Ohm's law for the direct current flow through the contact the electric current values can be expressed as

\[
I_{h} = h_{1b} g_{1b}  \\
I_{f} = h_{1y} g_{1y}
\]

Using the concept of the long constriction in the electric current flow \[2\], the classical solution of Hertzian contact between two cylinders and the penalty formulation for the normal contact defined for the point-wise contact in \[1\], the values of the electric conductance at the additional contact spots can be defined as

\[
h_{1b} = 2k \sqrt{\frac{3\varepsilon_{N} g_{N_{b}}}{4E}}  \\
h_{1y} = 2k \sqrt{\frac{3\varepsilon_{N} g_{N_{y}}}{4E}}
\]

where \(r, E\) and \(K\) are the equivalent values of: cross-section radius, Young's modulus and electric conductivity, respectively, computed for both contacting beams \[1\].

Thus, the electric current depends on both normal and voltage gaps, which is the source of semi coupling between mechanical and electric fields. Note, that in the classical penalty formulation for the normal contact used in this approach, the normal contact force \(F_{N}\) does not depend on the electric variables \(V\).

4. Governing equations and finite element discretisation

From the mathematical point of view the solution of a contact problem leads to the minimization of a functional subject to contact inequality constraints, i.e. a non-penetration condition. To solve this mathematical problem active set strategy can be used. The contact search routine yields the active constraints which considered the equality ones. In the presented three-point contact formulation there can be three active constraints – for the central pair and two additional pairs.

The global set of equations is obtained supplementing the global virtual work for two beams, by virtual work given by the contact contribution and analogous quantities for the electric field

\[
\Delta_{h} W_{M_{h}} = \Delta_{f} W_{M_{f}} + \Delta_{a} W_{M_{a}} + \Delta_{s} W_{M_{s}} = 0  \tag{8}
\]

The contact contributions are represented in \(8\) by the third components and can be defined as

\[
\Delta_{a} W_{M_{a}} = \bigcup_{active} (F_{N_{a}} \delta_{a} g_{N_{a}} + F_{N_{b}} \delta_{a} g_{N_{b}} + F_{N_{y}} \delta_{a} g_{N_{y}})
\]

\[
\Delta_{s} W_{M_{s}} = \bigcup_{active} (\Delta \delta_{s} g_{s} + I_{s} \delta_{s} g_{s} + I_{f} \delta_{s} g_{s})
\]

The non-linear set of equations \(8\) can be solved using the Newton-Raphson scheme, consistent linearisations with respect to the mechanical and electric variables are necessary. The appropriate expressions for the electric variables for all three possible contact pairs can be given as

\[
\Delta_{h} W_{M_{h}} = \Delta_{F_{N}} F_{N_{b}} \delta_{a} g_{N_{b}} + F_{N_{b}} \Delta_{F_{N}} \delta_{a} g_{N_{b}} + \Delta_{F_{N}} F_{N_{b}} \delta_{a} g_{N_{b}}  \\
+ F_{N_{b}} \Delta_{F_{N}} \delta_{a} g_{N_{b}} + \Delta_{F_{N}} F_{N_{b}} \delta_{a} g_{N_{y}} + F_{N_{y}} \Delta_{F_{N}} \delta_{a} g_{N_{y}}
\]

\[
\Delta_{a} W_{M_{a}} = \Delta_{a} F_{N_{a}} \delta_{a} g_{N_{a}} + \Delta_{F_{N}} F_{N_{a}} \delta_{a} g_{N_{a}} + \Delta_{F_{N}} F_{N_{a}} \delta_{a} g_{N_{a}}  \\
+ \Delta_{F_{N}} F_{N_{a}} \delta_{a} g_{N_{a}} + \Delta_{F_{N}} F_{N_{a}} \delta_{a} g_{N_{a}} + \Delta_{F_{N}} F_{N_{a}} \delta_{a} g_{N_{y}}
\]

5. Final remarks

Due to the fact, that mechanical contact variables do not depend on the electric field, the value \(\Delta_{h} W_{M_{h}}\) equals to zero. Thus, the FE discretisation leads to a semi-coupled problem with a non-symmetric contact tangent stiffness matrix, and the fully populated residual vector consisting of the mechanical and electric part, similarly as in the point-wise formulation given in \[1\].

References


Non-smooth dynamic viscoelastic frictional contact problem with memory modeled by hemivariational inequality

Stanisław Migórski
Faculty of Mathematics and Computer Science, Jagiellonian University in Krakow, ul. Lojasiewicza 6, 30-348 Krakow, Poland
e-mail: stanislaw.migorski@uj.edu.pl

Abstract

We consider an initial boundary value problem which describes the evolution of a viscoelastic body in contact with a piston or a device. In the contact problem the process is assumed to be dynamic and the friction is described with a Clarke subdifferential boundary condition of a locally Lipschitz nonconvex superpotential. The constitutive law is a generalization of the Kelvin-Voigt equation and involves a memory term. We derive a variational formulation of the contact problem which is in the form of a hemivariational inequality of a hyperbolic type. Then, we prove the existence of a weak solution and, under additional assumptions, its uniqueness. The proofs are based on recent results for history-dependent hemivariational inequalities and evolutionary inclusions in Banach spaces.

Keywords: Nonlinear inclusion, viscoelastic, friction, Clarke subdifferential, hemivariational inequality, normal compliance.

Introduction

In the paper we consider a mathematical model, which describes the dynamic process of contact between a viscoelastic body and a foundation. We are interested in the following physical setting. A viscoelastic body occupies a regular domain $\Omega$ of $\mathbb{R}^d$, $d = 2, 3$, with a Lipschitz boundary $\partial \Omega = \Gamma$. We assume that $\Gamma$ is composed of three sets $\Gamma_D$, $\Gamma_N$, and $\Gamma_C$ with mutually disjoint relatively open sets $\Gamma_D$, $\Gamma_N$, and $\Gamma_C$ such that $\text{meas}(\Gamma_D) > 0$. Let $[0, T]$ denote the finite time interval of interest, $Q = \Omega \times (0, T)$, $\Sigma_D = \Gamma_D \times (0, T)$, $\Sigma_N = \Gamma_N \times (0, T)$ and $\Sigma_C = \Gamma_C \times (0, T)$. The body is clamped on $\Gamma_D$. A volume force of density $f_0$ act in $\Omega$ and a surface traction of density $f_N$ act on $\Gamma_N$. The body is in contact on $\Gamma_C$ with an obstacle, the so-called foundation. We denote by $u = (u_i)$, $\sigma = (\sigma_{ij})$, and $\varepsilon(u) = (\varepsilon_{ij}(u))$, $i, j = 1, \ldots, d$, the displacement vector, the stress tensor, and linearized strain tensor, respectively. We recall that the components of the linearized strain tensor $\varepsilon(u)$ are given by

$$
\varepsilon_{ij}(u) = \frac{1}{2}(u_{i,j} + u_{j,i}).
$$

For a vector field $v$, we use the notation $v_\nu$, $v_r$ for the normal and tangential components of $v$ on $\Gamma$ given by $v_\nu = v \cdot \nu$ and $v_r = v - v_\nu \nu$. We recall that the normal and tangential components of the stress field $\sigma$ on the boundary are defined by $\sigma_\nu = (\sigma u) \cdot \nu$ and $\sigma_r = \sigma u - \sigma_\nu \nu$. We also use the symbol $\mathbb{S}^d$ for the space of second order symmetric tensors on $\mathbb{R}^d$ or, equivalently, the space of symmetric matrices of order $d$.

We consider a viscoelastic body which is attached to a piston or a device over the surface $\Gamma_C$. Following the terminology used in Contact Mechanics we refer to $\Gamma_C$ as the contact surface and, below, the device will be referred as the obstacle or the foundation. Note that, according to the physical setting, no separation between the body and the obstacle is allowed, which represents one of the novelties of the model we introduce below. Then, the classical formulation of the problem is as follows.

Problem $P$: Find a displacement field $u: Q \rightarrow \mathbb{R}^d$ and a stress field $\sigma: Q \rightarrow \mathbb{S}^d$ such that

$$
\sigma(t) = A(t, \varepsilon(u'(t))) + B(t, \varepsilon(u(t)))
+ \int_0^t C(t - s)\varepsilon(u'(s)) \, ds \quad \text{in } Q, \tag{1}
$$

$$
u''(t) = \text{Div} \sigma(t) + f_0(t) \quad \text{in } Q, \tag{2}
$$

$$
u(t) = 0 \quad \text{on } \Sigma_D, \tag{3}
$$

$$
\sigma(t) \nu = f_N(t) \quad \text{on } \Sigma_N, \tag{4}
$$

$$
-\sigma_r(t) = p(t, u_r(t)) \quad \text{on } \Sigma_C, \tag{5}
$$

$$
-\sigma_\nu(t) \in \partial \psi(t, u_\nu(t)) \quad \text{on } \Sigma_C, \tag{6}
$$

$$
u(0) = u_0, \quad \nu'(0) = v_0 \quad \text{in } \Omega. \tag{7}
$$

Equation (1) represents the viscoelastic constitutive law in which $A$ is a nonlinear operator which describes the viscous properties of the material, $B$ is a nonlinear operator which describes its elastic behavior, and $C$ represents the relaxation tensor. Various results, examples and mechanical interpretations in the study of such kind of constitutive laws can be found in [1] and the references therein. Such kind of laws were used in the literature in order to model the behavior of real materials like rubbers, rocks, metals, pastes and polymers. In particular, equation (1) was employed to model the hysteresis damping in elastomers. Note that in the case when the relaxation tensor vanishes and the viscosity and elasticity operators are time-independent, equation (1) is reduced to the Kelvin-Voigt constitutive equation.

Equation (2) represents the equation of motion in which, for simplicity, we supposed that the mass density is equal to one. Conditions (3) and (4) are the displacement and the traction boundary conditions, respectively. They model the situation when the body is fixed on the part $\Gamma_D$ of its boundary and the Cauchy stress vector is prescribed on $\Gamma_N$, respectively.

Equation (5) is the normal compliance contact condition in which $\sigma_\nu$ denotes the normal stress, $u_\nu$ is the normal displacement, and $p$ is a given function which describes the instantaneous reaction of the obstacle. The reaction of the obstacle depends
both on the current value of the normal displacement represented by the term \( p(t, u_n(t)) \).

Condition (6) represents friction law, where \( j_r \) is a given locally Lipschitz, in general, nonconvex and nonsmooth function, and symbol \( \partial j_r \) denotes the Clarke subdifferential of \( j_r \) with respect to its last variable. Since the function \( j_r \) can be nonconvex, the friction law is generally nonmonotone and of zig-zag type. Concrete examples of frictional conditions which lead to subdifferential boundary conditions of the form (6) can be found in [3, 4]. Here, we only remark that these examples include the nonmonotone friction law, the power-law friction, the Tresca friction law as well as its regularization.

Finally, (7) represents the initial conditions in which \( u_0 \) and \( v_0 \) denote the initial displacement and the initial velocity, respectively.

In order to provide the variational formulation of Problem \( P \), we recall the following definitions and notation. Given \( h: E \to \mathbb{R} \) be a locally Lipschitz function defined on a Banach space \( E \), the generalized directional derivative of \( h \) at \( x \in E \) in the direction \( v \in E \), denoted by \( h^\circ(x; v) \), is defined by

\[
\lim_{y \to x; \lambda \to 0} \frac{h(y + \lambda v) - h(y)}{\lambda}
\]

and the generalized gradient of \( h \) at \( x \), denoted by \( \partial h(x) \), is a subset of a dual space \( E^* \) given by

\[
\partial h(x) = \{ \zeta \in E^* | h^\circ(x; \zeta) \geq \zeta, \forall \zeta \in E \}.
\]

We introduce the spaces \( V \) and \( H \) by

\[
V = \{ v = (v_t) \in H^2(\Omega; \mathbb{R}^n) | v = 0 \text{ a.e. on } \Gamma_D \},
\]

\[
H = \{ \tau = (\tau_{ij}) | \tau_{ij} = \tau_{ji} \in L^2(\Omega), 1 \leq i, j \leq d \}.
\]

The duality pairing of \( V^* \) and \( V \), and the inner product in \( H \) are denoted by \( \langle \cdot, \cdot \rangle_{V^* \times V} \) and \( \langle \cdot, \cdot \rangle_H \), respectively. Moreover, we set

\[
V^* = L^2(\Omega, T; V^*),
\]

where \( V^* = L^2(\Omega, T; V^* \times V^*) \) is the dual space to \( V \) and the time derivative \( w^t = \partial w/\partial t \) is understood in the sense of vector-valued distributions. Also, we introduce the trace operator \( \gamma : V \to L^2(\Gamma, \mathbb{R}^n) \). Under these notations, the variational formulation of Problem \( P \) reads as follows.

**Problem \( P^V \).** Find a displacement field \( u : (0, T) \to V \) and a stress field \( \sigma : (0, T) \to H \) such that

\[
\sigma(t) = A(t, \varepsilon(u(t))) + B(t, \varepsilon(u(t))) + \int_{\Gamma_C} \mathcal{C}(t-s)\varepsilon(u^*(s)) \, ds
\]

for a.e. \( t \in (0, T) \),

\[
\langle u^*(t), v \rangle_{V^* \times V} + \langle \sigma(t), \varepsilon(v) \rangle_H + \int_{\Gamma_C} p(t, u_n(t))v_n \, d\Gamma
\]

for all \( v \in V \), a.e. \( t \in (0, T) \),

\[
u(0) = u_0, \quad u'(0) = v_0.
\]

From Problem \( P^V \) we obtain the following abstract formulation of the contact problem which is called a hemivariational inequality.

**Problem \( HV I \).** Find \( u \in V \) such that \( u^* \in W \) and

\[
\langle u''(t) + A(t, u'(t)) + \mathcal{S}u'(t), v \rangle_{V^* \times V} + \int_{\Gamma_C} j^0(t, u(\gamma_n(t)); v) \, d\Gamma \geq \langle f(t), v \rangle_{V^* \times V}
\]

for all \( v \in V \) and a.e. \( t \in (0, T) \),

\[
u(0) = u_0, \quad u'(0) = v_0.
\]

where \( A : (0, T) \times V \to V^* \), \( \mathcal{S} : V \to V^* \) and \( j : \Sigma_C \times \mathbb{R}^d \to \mathbb{R} \) are defined by

\[
\langle A(t, u), v \rangle_{V^* \times V} = \langle A(t, \varepsilon(u)), \varepsilon(v) \rangle_H
\]

for all \( u, v \in V \), a.e. \( t \in (0, T) \),

\[
\langle \mathcal{S}u'(t), v \rangle_{V^* \times V} = \sum_{i=1}^3 \langle (\mathcal{S}_i u'(t), v) \rangle_{V^* \times V}
\]

for all \( u \in V \), \( v \in V \), a.e. \( t \in (0, T) \), with

\[
\langle \mathcal{S}_1 u'(t), v \rangle_{V^* \times V} = \langle B(t, \varepsilon(\zeta w(t))), \varepsilon(v) \rangle_H
\]

\[
\langle \mathcal{S}_2 u'(t), v \rangle_{V^* \times V} = \int_{\Gamma_C} \left( \mathcal{C}(t-s)\varepsilon(w(s)) \right) w_n \, d\Gamma
\]

\[
\langle \mathcal{S}_3 u'(t), v \rangle_{V^* \times V} = \int_{\Gamma_C} \mathcal{P}(t, \zeta_0 w(t)) \, v_n \, d\Gamma
\]

for all \( w \in V \), \( v \in V \), a.e. \( t \in (0, T) \), and

\[
j(x, t, \xi) = j_r(x, t, \xi, \gamma) \quad \text{for all } \xi \in \mathbb{R}^d, \text{ a.e. } (x, t) \in \Sigma_C.
\]

In the definitions of operators \( \mathcal{S}_1 \) and \( \mathcal{S}_3 \), we use the following notation

\[
\zeta w(t) = \int_0^t w(s) \, ds + u_0
\]

for all \( w \in V \) and \( t \in (0, T) \), and

\[
\zeta_0 w(t) = \zeta w(t)|_{t=0}^T = \int_0^T w(s) \, ds + u_0.
\]

For the above hemivariational inequality, Problem \( HV I \), we provide a result on its unique weak solvability. The proof is based on recent results for history-dependent hemivariational inequalities and evolutionary inclusions in Banach spaces, cf. [2, 4, 5].

In conclusion, we deduce that the frictional contact problem for a viscoelastic body with memory term admits a unique weak solution. Finally, we underline that through the formulation of hemivariational inequalities, problems involving nonmonotone and multivalued constitutive laws and boundary conditions can be treated successfully mathematically and numerically. From this point of view, the inequality problems in mechanics can be divided into two main classes: that of variational inequalities, which is concerned with convex energy functions (potentials), and that of hemivariational inequalities, which is concerned with nonsmooth and nonconvex energy functions (superpotentials).

**References**


Rolling contact problems for plastically graded materials

Andrzej Myśliński\textsuperscript{1}, Andrzej Chudzikiewicz\textsuperscript{2}
\textsuperscript{1}Faculty of Manufacturing Engineering, Warsaw University of Technology
ul. Narbutta 85, 02-542 Warsaw
e-mail: myslinski@ibspan.waw.pl
\textsuperscript{2}Institute of Transport, Warsaw University of Technology
ul. Koszykowa 75, 00-662 Warsaw
e-mail: ach@it.pw.edu.pl

Abstract

The paper deals with the numerical solution of thermo-elastoplastic wheel-rail rolling contact problems including friction, frictional heat generation and transport across contact surface. Two-material rail model is assumed, where Young’s modulus of a rail material near the rail surface is dependent on depth of the rail. The heat flow is governed by the heat conduction equation. The equilibrium state of this contact problem is described by the coupled hyperbolic variational inequality of the second order and a parabolic equation. Numerical examples are provided and discussed.

Keywords: Elasto-plastic rolling contact problem, frictional heat generation, finite element method

1. Introduction

The paper deals with a numerical solution of the wheel-rail contact problems including friction and frictional heat generation. The contact of a rigid wheel with an rail lying on a rigid foundation is considered. The friction between the bodies is governed by the Coulomb law [6, 9, 10]. The elastic or elastoplastic rolling contact problems were considered by many authors. For details see references [1, 2, 5, 6, 9, 10, 11, 12, 13, 14]. Numerical solutions to assess partially plastic and fully plastic deformation behaviour of a functionally graded spherical pressure vessel are presented in [1]. The material modulus of elasticity is governed by the power law in normal direction. Tresca’s yield criterion is used to indicate plastic regions for the graded material. Thermoplastic instability in two dimensional contact problem is considered in [2] where the heat flow in friction material components is described by conduction–convection term and a standard elastic contact formulation is used. $J_2$ plasticity model and an isotropic hardening law are assumed. By means of the finite element method the thermal and elastoplastic contact problems are solved sequentially in time. A comprehensive parametric study of the mechanics of normal indentation of plastically graded materials is performed in [5] using finite element method and dimensional analysis. The yield strength of the layers is assumed to depend on the surface yield strength as well as on the depth of the material. The elastic and plastic responses are approximated by Hooke’s and von Mises yield criterion with isotropic power law hardening. From simulations universal dimensionless functions are extracted to predict the indentation load versus depth of penetration curves for a wide range of materials. The paper [13] deals with the derivation of the contact stress in wheel-rail system consistent with a perfect plastic law. Assuming a perfect plastic behaviour for the wheel and rail multi-Hertzian approach is extended and the formulas for the pressure and the length of the ellipses semi-axes are provided. The comparison of the proposed method to the finite element method and other Hertzian methods is derived.

Numerous laboratory or numerical experiments [11] indicate that the use of a coating functionally graded material attached to the conventional steel body reduce the magnitude of residual or thermal stresses and rolling contact fatigue. Functionally graded materials are multiphase composites mainly composed of a ceramic and a metal. They exploit the heat, oxidation and corrosion resistance typical of ceramics, and the strength, ductility and toughness typical of metals. In a conventional coating structure homogeneous materials are used. The abrupt change in the mechanical or thermal properties of the materials at the surface coating-substrate interface results in stress concentration or degraded bonding strength.

Therefore in the paper we solve numerically the wheel-rail contact problem with friction assuming plastically graded model of the coating layer rather than elastic as in [6]. In the paper the time-dependent model of this rolling contact problem is introduced. The elastic and plastic responses are approximated by the Hooke’s law and von Mises yield criterion with isotropic power law of hardening, respectively [4, 5]. Finite element method is used as a discretization method. It is well known that the application of the classical finite element method, where material properties are constant, to solution of problems with the functionally graded materials may lead to large numerical errors. A proper approach to solve such problems requires application of nonhomogeneous finite element method containing additional approximation functions in order to interpolate material properties at the level of each finite element. This idea is implemented in the framework of the graded [8] or multi-scale [7] finite element methods. The distribution of stresses including the normal and tangent contact stresses as well as the distribution of the temperature are numerically calculated. The provided results are discussed.

2. Problem Formulation

Consider deformations of a two-dimensional strip lying on a rigid foundation. A wheel rolls along the upper surface of the
strip. The axis of the wheel is moving along a straight line at a constant altitude and the wheel is pressed in the elastoplastic strip. It is assumed, that the head and tail ends of the strip are clamped, i.e., we assume that the length of the strip is much bigger than the radius of the wheel. Moreover it is assumed, that there are no mass forces in the strip. The strip consists from two layers. The mechanical and thermal properties of the surface layer are assumed vary throughout the material. Especially the modulus of elasticity is assumed to be governed by the power law in the vertical direction. The Poisson’s ratio is assumed constant. A conventional model of isotropic plasticity is assumed. The yield function depends on effective Mises stress, the yield stress on the upper surface and the isotropic hardening function.

3. The thermo-elstoplastic model

Since the rolling contact problem involves thermomechanical coupling it requires to develop separate algorithms for the thermal and elastoplastic analysis. The heat flow is governed in both layers by the heat conduction equations. The heat generated by the plastic deformation is assumed to be suitably small. The boundary conditions associated with the heat conduction are due to the contact between layers and the surrounding environment. The contact between the layers generates heat flow due to friction, and the frictional heat flux is proportional to the friction coefficient, the contact pressure and the sliding velocity. For details see [6]. A standard contact formulation is used imposing the continuity of normal displacement across the contact interfaces if the contact condition is satisfied.

![Figure 1: Normal contact distribution for different nonhomogeneity parameters α](image)

4. Numerical implementation

The finite element method is used as discretization method. In the numerical procedure the thermal and elastoplastic contact problems are solved sequentially in time. For the contact pressure calculated in the previous time step the heat flux and the temperature are evaluated for the next time step. If the temperature increment is less than the prescribed value the algorithm is stopped. Otherwise the contact pressure for the next time step is evaluated. Here we use semi-smooth Newton method to solve the elastoplastic contact problem. This approach consists in reformulating the complementarity conditions as a set of semi-smooth equations.

5. Conclusions

The obtained numerical results indicate that the obtained contact patches are characterized by longer zones and lower stress intensity than in the elastic case. Moreover the graded layer can reduce the values of the normal contact stress (see Fig. 1) and the maximal temperature in the contact zone. The obtained stress and temperature distributions are dependent on the nonhomogeneity index. For higher values of it one can obtain the higher differences in the maximal contact stress and temperature in comparison to the homogeneous material case. In order to improve the model one could consider finite strain plasticity. The choice of the depth of the coating layer may be based on the optimization method to ensure the optimal reduction of the normal contact stress.

References


Analysis of contact pressures in embossing process of regular asperities of surface

Radosław Patyk¹, Łukasz Bohdal²

¹,² Faculty of Mechanical Engineering, Koszalin University of Technology
Raclawicka 15-17, 75-638 Koszalin, Poland

Abstract

Burnishing as a metal finishing treatment allows to receive product with a high smoothness of surfaces and advantageous stress distributions. The problem still unsolved, connected with a design of burnishing rolling process is the calculation of the main shaping force, which is the most important technological parameter, determining outline changes of the deformed surface. The values of the forces are calculated on the basis of formulas from literature and they are quite different. In the paper the analysis of the contact pressure and the outline changes of the deformed surface which appear during plastic shaping of regular asperities with rigid tool are presented. The known way of analytical solution of this problem using a characteristic method is presented with the following simplifications: lack of friction force in the contact zone, no elastic deformations of the deformed body, no strengthening (ideal rigid – plastic body model). Such an analytical solution can be randomly used in industrial practice only. In order to eliminate these inaccuracies, two new methods of analysis are proposed (theoretical and numerical), based on the Finite Element Method (FEM), taking into account real conditions of process realization. In the numerical method additional elastic strains are considered. For this case an application in ANSYS/LS-DYNA was elaborated, which allows to make the calculation for the materials capturing linear and nonlinear strengthening, strain rate and temperature. The exemplary calculations using the proposed method are presented, assuming the same geometry of asperity, but different mechanical characteristic of the deformed material.

Keywords: burnishing process, embossing process, burnishing forces, contact pressures, finite element method

1. Introduction

Embossing of regular surface asperities is one of the basic processes analysed in plastic treatment of solids. In the developed solutions of better understanding, the process of cutting or pressing are aimed. Some of these studies attempt to adopt the processes of forming treads with different outlines or specific treatment consisting of plastic shaping regular asperities, which are next smoothly burnished (it is called duplex burnishing). This treatment is designed to significantly improve the quality of products and to control distribution and value of the residual stresses. A crucial problem in the development of technological processes is the knowledge of the technological parameters and the impact of their variation on the technological quality. In the embossing rolling process the basic technological parameters are: burnishing speed, burnishing feed and force or depth of burnishing [1,2,3,6,7]. The most difficult task is to determine an appropriate value of the force or interference in burnishing which are intended to produce the required effects, e.g. to improve the surface condition and constitution of the relevant parts as well as the state of stress in the Surface Layer (SL) product. The resultant burnishing force is the geometrical sum of its components: F₁ - tangential force, F₂ - axial (longitudinal) and F₃ - normal force called the main burnishing force.

\[
F = \sqrt{F_1^2 + F_2^2 + F_3^2}
\]  

An improperly used burnishing force can be the reason of not receiving a required (large) surface smooth or spalling. Therefore a technological parameter of the process is determined, which is responsible for a utilized quality. The component of a burnishing force Fᵢ (i=1, 2, 3) can be calculated from the relation:

\[
F_i = \sum_{k=1}^{n} F^k_i
\]  

where, \( F^k_i \) is the component of the burnishing force falling on the k asperity of the surface, in the P position of the tool, \( n \) is number of asperities which are plastically deformed.

The force \( F^P_k \) is described by:

\[
F^P_k = \iiint_D q^P_k d\Omega^P_k
\]  

where: \( q^P_k \) is elementary resultant force acting on asperity at research point, \( \Omega^P_k \) the projection of region \( \Sigma^P_k \) on the plane \( y_1, y_2, \Sigma_k^P \) contact zone of the tool with the object (Fig. 1).

![Figure 1: Forces acting on elementary region dS.](image)

The calculation of burnishing force components from the formula (3) is possible only for the case, when the object material is regarded ideally rigid-plastic. For other cases, when material is hardened, analytical calculation of the integrals from the formula (3) is impossible. Then value of the main burnishing force is calculating numerically. In the paper the trial of calculation of pressure value and distribution in contact zone for the case of embossed regular asperities was undertaken. The contact pressure causes displacement of a body boundary in the contact. The
value of this displacement depends on contact stiffness, defined by relation of the surface force divided by the value of the displacement of the contact area in a direction of the force. There are various normal and tangential contact stiffnesses. The relation of elementary force and displacement \((p_i = f(u_i))\) can be approximated by means of two methods.

2. Theoretical analysis

A wedge about vertical angle \(2\theta\) pressed in surface is considered. Analytical solution of the issue using unit plane according to characteristic method was conducted \([4,5]\). The pressure in the contact zone is calculated from equation:

\[ p = 2k(1 + \gamma), \]

and a force required for a wedge press from the relation:

\[ P = 4 \cdot k \cdot a \cdot s \cdot (1 + \gamma) \cdot \sin(\theta), \]

where: \(s\) is wedge width, \(\gamma\) is angle of polar sector and \(a\) is length of the side, \(k\) is material parameter (it is a value of yield stress for ideal shear).

METHOD I – constant stress

In the literature solution nonlinear plasticity condition by Huber-Mises-Hencky is accepted, described by the equation:

\[ k = \frac{R_e}{\sqrt{3}}, \]

where, \(R_e\) is material yield stress. Then the pressure \(p_e\) depends on the geometry of the wedge and the material yield stress. However, advancement does not depend on the degree of deformation, so \(p_e = p = \text{const}\). This solution applies only to perfectly rigid-plastic materials, without hardening.

METHOD II (Author’s modification – variable stress)

In the second solution in order to calculate \(k\) a nonlinear plastic Huber-Mises-Hencky condition was used, whose variant for the hardened material uses the stress:

\[ k = \frac{\sigma_0(\varepsilon_i)}{\sqrt{3}}, \]

where \(\sigma_0(\varepsilon_i) = R_e(\varepsilon_0 + \varepsilon_i)^n\) is material plastic stress (stress as the function of strain).

The presented solution has limited applications in engineering practice because complex contact zones occur between the tool and treated object and it does not include possibility of free material flow blocking. The better solution is given by application of Finite Element Method to analysis of contact pressure distribution in plastic treatment process.

3. Numerical analysis

For the pressure analysis in the contact zone between regular asperity embossing two computer models of the process using the finite element method were developed. The first model is the simplest method of forming regular asperities where there is no free material movement blocking phenomenon in forming asperity. In the second model, the free material movement blocking phenomenon of forming regular asperity does not occur (Fig. 2). The results of the conducted analysis are presented in the graph (Fig. 3.)

![Figure 2: Computer models of the embossing process of regular asperities a) model I, b) model II](image)

![Figure 3: Relation of contact pressure as a strain function for the theoretical and numerical solutions](image)

4. Conclusions

The solution presented in literature \([4,5,7]\) has a limited use and can be applied for materials with the free movement of the material.

Theoretical solutions presented by authors (modification of Hill solution) can be used for materials with hardening, but only in the range of not too large plastic strains.

The proposed numerical solutions show a much better compatibility with reality in relation to the theoretical solutions and is possible to be applied in engineering practice. These solutions can contribute to the optimization of tools to reduce and to change the distribution of pressure in the contact zone, in order to greatly improve durability.

5. References

Analysis of numerical model parameters of a modular scaffolding node loaded with shearing force

Michał Pieńko¹, Ewa Blazik-Borowa²

¹,² Lublin University of Technology, Faculty of Civil Engineering and Architecture, Department of Structure Mechanics
Nadbystrzycka 40 street, 20-618 Lublin, Poland
e-mail: m.piengo@pollub.pl¹; e.blazik@pollub.pl²

Abstract

The purpose of the article is to present issues related to the modeling of a modular scaffolding node. The node loaded with shearing force was subjected to the analysis. Due to the nature of work and complexity of the construction, it is necessary to take account of the phenomena associated with friction and contact. A proper numerical modeling of a node requires laboratory testing to track the behavior of the node as well as information of the material characteristics of the node elements. The numerical model accurately mapping behavior of the node, allows applying force to the nodes in any ways and any number of elements (max 8). This is an alternative method for determining the capacity of the nodes in relation to more expensive laboratory tests.

Keywords: nonlinear numerical calculation, scaffolding node, friction, contact, plastic deformations

1. Introduction

In the majority of publications concerning scaffoldings, collected and discussed in [2], the authors consider the scaffolding as the whole construction. One of the most common issues, undertaken by the authors, is the subject referring to the stability of the scaffold used as a support to boarding [3, 6]. The issue of bamboo scaffolds has been widely described. One of the papers [1] discusses the numerical analysis as well as the laboratory tests of a steel node as an alternative solution to join particular elements of scaffolding. Moreover, the information on junction characteristics of scaffolding elements, which can be applied to commercial modelling programs for scaffold constructions, can be found in the paper [4]. The authors of the work [5] were the only to attempt to create a solid model of numerical modelling of a modular scaffolding node. Hence, the article demonstrates the subsequent steps of numerical model creation, taking into account shear force load of a node. Due to the lack of direct availability to the points where plastic deformation takes place, it is necessary to perform a number of numerical analysis variants that thoroughly consider the behaviour of nodes affected by the investigated load. In the paper, the impact of two parameters of node geometry (the depth of wedge drive and end rotation) on the errors of obtained results was analyzed.

2. Laboratory tests

Laboratory tests were preceded by a development of the grip handle to allow loading the node in the testing machine (Fig. 1a). In order to minimize the influence of bending on the deformation form affected by shearing in the proposed holder, the arm of an acting force was minimalized and massive clamps were perfectly matched to the diameter of ledger pipe. The scheme of loading was reversed and shearing force in the ledger was induced by displacement of the pipe stand. As a result of conducted laboratory tests, the graph presenting the dependence of displacement and shearing force was obtained. Summarizing, five attempts at testing were carried out, in every case the same form of damage was received in the form of ledger end split (Fig. 1b). In order to determine material characteristics of various elements of the node, samples were taken from the scaffolding. The characteristics were obtained from static tensile tests on five flat samples. Unfortunately, each element of a node is made of the material of different strength characteristics.

Figure 1: Laboratory test of shearing force a) grip b) deformation of the ledger end

3. The numerical analysis

The numerical analysis of nodes in a modular scaffolding was started by collecting all the necessary dimensions of the individual parts of the nodes and their material characteristics.

3.1. Creating FEA models

The creation of a detailed model was possible due to the fact that technical drawings of node elements (Fig. 2a) were made available by the manufacturer. Due to symmetry of the system subjected to the research in the numerical model, only the half of the system was analyzed. Numerical analysis was conducted in the ABAQUS program which easily introduces contact between particular elements. This program made it possible to build FEM element mesh. The C3D8R elements were applied to the majority of node elements. In the case of ledger end, in the areas of neither plasticization nor contact, the C3D4 elements were used due to the complicated geometry. The numerical model contained 75912 elements and 61520 nodes. In welds occurring between the ledger end and ledger pipe, full junction was applied. Owing to the fact that the welds joining the rosette with ledger pipe occur in a sensitive place in which loading is transmitted, they were modelled as solid elements. In the case of other undurable elements, the type of hard contact and fric-
tion with the coefficient $\mu = 0.15$ were applied. This type of contact was found between the wedge and end, end and rosette, rosette and wedge. Since even at small displacements, local plasticization can occur in a model, a linearly reinforced material was used. Material characteristics were obtained from laboratory tests on particular elements of nodes. Due to large deformations, apart from nonlinearities resulting from the use of contact, geometrical nonlinearities were also taken into account.

![Figure 2: Numerical model of the node with ledger a) geometry b) rotation of the ledger end c) boundary conditions d) inserting the wedge e) shearing the ledger](image)

The process of nodal loading was divided into two stages. In the first stage, the wedge was driven into the hole of a rosette. Due to the lack of accurate control of the wedge immersion in the hole during lab tests, the analyses with three variants of wedge drive 0mm, 1mm, and 2mm were performed. The driving of a wedge induced local plasticization of a rosette within the hole. The subsequent step of loading was the displacement of a pipe in a massive clamp of a testing machine. Significant discrepancies between the results of numerical and laboratory analyses motivated for the use of another parameter the rotation at the end. Three cases of inaccuracy of node execution or assembly were considered; 0°, 0.75° and 1.5° (Fig. 2b). As the parameter defining the compatibility of the obtained results, the percentage error dimension calculated as the ratio between the difference between diagram areas obtained in the laboratory research and numerical analyses according to the formula:

$$\Delta = \frac{\int (F_{d} - F_{l})du_{z}}{\int F_{d}du_{z}},$$

where: $F_{d}$ – force function of the force in numerical analysis; $F_{l}$ – force function of the force in laboratory tests; $u_{z}$ - displacement.

The error values are shown in Table 1. The present difference of a value 6.3% was obtained in the case of ledger end rotation by 1.5° and the driving of a ledger to the depth.

![Figure 3: The graph of the displacement ($u_{z}$) in the node caused by shearing force ($V_{z}$) applied to the ledger](image)

As seen in the graph, the driving of a wedge does not affect the shape of a graph. The difference resulting from the depth of wedge driving is maximum 2.04%, and it occurred in the case of the rotation by 0.75°.

### 4. Summary

Due to the performed variants of numerical analysis the information on the sensitive node elements was obtained, with an impact on the obtained results of laboratory tests. In the case of a node loaded by a shear force, the imperfections resulting from the construction of elements and inaccuracy of node assembly act strongly on the obtained results. In the case of this type of loading, the depth of wedge driving does not affect the results. The created numerical model can be used to load the node with a greater number of elements without the necessity to carry out expensive and complicated laboratory tests. Further research of the node requires the investigation into the impact of parameters associated with the phenomenon of friction.

### References


FEM simulation of electromagnetic forces in the rails of electromagnetic launcher

Justinas Račkauskas¹, Rimantas Kačianauskas², Markus Schneider³

¹,² Department of Strength of Materials and Engineering Mechanics, Vilnius Gediminas Technical University
Saulėtekio 11, 10223 Vilnius, Lithuania
e-mail: justinas.rackauskas@vgtu.lt, rimantas.kacianauskas@vgtu.lt²
³ French-German Research Institute of Saint Louis (ISL)
5 rue G’al Cassagnou, 68301 Saint-Louis, France
e-mail: markus.schneider@isl.eu

Abstract

The current flowing in current conducting materials induces the electromagnetic forces. The paper describes the finite element analysis of a linear electromagnetic launcher. The demonstrated distribution of the volume forces presents the results obtained in studying the distribution of forces along the rail during launching. The calculations made with operating of a fixed projectile position shows that the force is distributed non-uniformly along the rail in the case of the rail-armature operation. The simulation results are compared to the solutions obtained using a simplified engineering approach. The main differences in distribution of forces in the rail in the contacting zone between the rail and the armature are discussed.

Keywords: volume forces, electromagnetic launcher, mechanical force

1. Introduction

The linear EM launcher is a mechatronic device aimed at accelerating small mass equipment and potentially offering the possibility to reach muzzle velocities higher than 2 km/s. The idea of linear Electromagnetic (EM) launcher was put forward in the 19th century. The first efforts to develop an EM launcher were undertaken by Kristian Birkeland in Norway [2]. Over the last 100 years, many experiments have been made in order to improve the launcher design and to create more powerful devices. The mechanical structure of the device presents two electric current-conducting rails and an armature between them [10]. The power supply equipment (e.g. a capacitor and a pulse forming unit) and other components (e.g. switches) are required in order to operate an EM launcher [7].

The French–German Research Institute of Saint Louis (ISL) operates various linear EM launchers now, such as PEGASUS [1], RAFIRA [3]. The extensive research on electromagnetic and mechanical behaviour of these devices was performed [5].

In the paper the results are presented of the finite element electromagnetic analysis of a linear electromagnetic launcher and the calculations of electromagnetic force distribution along the rail.

2. Description of a linear EM launcher

The setup of the linear EM launcher is presented in Fig. 1. It consists of two rails, two housing bars, insulators and structure fixing bolts. The housing bar, insulators and steel bolts maintain the current conducting rails in the fixed position.

Generally, a launcher is a facility comprising typical components, such as an accelerator and a projectile, including a conducting armature and an electric energy source. The electric circuit is formed by a power source, the rails and the sliding contact between the latter realized by the armatures. The pulse-forming unit generates the pulsed currents creating high (in the range of some Tesla) magnetic fields. As a result, the projectile is accelerated by the Lorentz-force. A negative aspect is that the railgun structure must be designed to withstand strong repelling Lorentz forces.

Figure 1: The setup of a linear electromagnetic launcher
A fragment of the limited length $L = 20$ cm and the EM launcher was modelled for the analysis only. It was sufficient to demonstrate a force distribution along the rail. The calculation model is presented in Fig. 2. The dashed lines indicate the symmetry lines. Due to the model symmetry it was sufficient to model one quarter of the whole EM launcher model. The red lines indicate the part which was modelled (the view from the back quarter of the armature and a half of rail). The modelled rail cross-section was $2.5 \times 2.0$ cm while the gap between the rails $w = 2.5$ cm. To save the calculation resources, the housing bars and steel bolts were not included in the calculation model.

Figure 2: A physical model of the launcher: $I$ indicates the flowing electric current, $F_L$ denotes the Lorenz force

Figure 2: A physical model of the launcher: $I$ indicates the flowing electric current, $F_L$ denotes the Lorenz force
3. Modelling

The electromagnetic finite element analysis comprises the numerical solution of Maxwell equations [9]: consisting of four equations: Ampere-Maxwell law, Faraday’s law, Gauss’ law and Gauss’s law for magnetism. The quantities such as charge density (\(\rho\)), electric field density (\(D\)), magnetic flux density (\(B\)), electric field intensity (\(E\)), magnetic field strength (\(H\)) and current density (\(J\)) were calculated.

The Lorentz force is the integral used to calculate the force in any current carrying volume (a finite element). It is result of cross product of current density and magnetic flux \(\mathbf{f} = (\mathbf{J} \times \mathbf{B})\) [9].

The induced Lorentz forces have two basic components. The component perpendicular to rail presents the repelling forces acting on the rails. These forces play an important role in the EM launcher technology because, due to high currents in the rails, pressure of several hundred MPa is created. As a result, the gap between the rail and the armature appears, and the current conducting properties in the contact zone between the rail and the armature are reduced.

In order to investigate force density, the simplified 3D finite element model of the conducting part (one quarter of the armature and a half of rail) of EM launcher was developed. This model was built and calculated based on the ANSYS software code [8], operating using FEM.

4. A brief description of the results of force density

For the first calculation the fixed position mode of the EM launcher operation was simulated. The high frequency harmonics of 1250 Hz with current pulse peak of 650 kA were used. Two different cases of the EM launcher operation modes were considered for demonstrating the repelling force distribution along the rail. The results of the FE calculations are presented in Fig. 3, where the variation of the repelling force along the rail on the inner surface edge is shown by thin solid lines. The results demonstrate high gradients in the vicinity of the projectile, where the current density vector is changing its direction.

![Figure 3: The variation of the repelling force density along the inner surface edge of the rail: Case 1 and Case 2 present the FE analysis, a black line denotes the engineering approach](image)

In case 1, representing the model, consisting of the rail and the armature, the electric current changed the direction of flowing, and the forces distributed non-uniformly in the region near the rail-armature contact zone [6]. Another result presented in case 2 of the model, consisting of rails only, the current flows along the rail and forces act on the rail uniformly (green). In comparison, the black straight line shows the load force density calculated using the engineering method, commonly employed in mechanical investigation of EM launchers and described in [4]. The engineering method is related with high frequency approximation. This type of the force load is distributed uniformly on all rail surfaces. The mechanical load is used to obtain constant pressure on the inner surface.

5. Conclusion

Numerical results obtained in the performed electromagnetic FE analysis represent the important features of the mechanical repelling forces acting along the rails of an electromagnetic launcher. It was confirmed that uniform variation of the repelling force acting on the rail, recently used for mechanical modelling of launcher dynamics are not able for the mechanical load in the vicinity of the projectile. The local effect demonstrates the increased force gradient in back of the projectile and the reduced pressure in \(y\) direction in the contact zone.

References

A study of surface roughness in elasto-plastic shrink-fit assembly

Arkadiusz Rzeczycki¹, Ryszard Buczkowski²
¹,² Department of Computer Methods, Maritime University of Szczecin
ul. Pobożnego 11, 70-507 Szczecin, Poland
e-mail: a.rzeczycki@am.szczecin.pl¹, rbuczkowski@ps.pl²

Abstract

The normal contact stiffness obtained from fractal and finite element models was incorporated into the finite element system Abaqus via a subroutine UINTER to analyze a shrink-fit assembly. Calculations were carried out for different machining processes.

Keywords: contact stiffness, rough surfaces, fractal model

1. Introduction

Mechanical joints are widely used in machines and mechanical structures. They are highly dependent on the roughness of contacting surfaces. A contact phenomenon of joints is complex and highly nonlinear.

Modelling of rough surfaces contact was modelled using a number of approaches. Conventionally, the deviation from the mean plane of the surface is assumed to be a random process and is characterized by statistical parameters: standard deviation of the surface height \( \sigma \), slope \( \sigma' \) and curvature \( \sigma'' \) [1]. Theories of elastic contact introduced by Greenwood and Williamson [3] and plastic contact proposed by Nayak [1] use these parameters. However, when these parameters are measured in one scale, they change completely with the change of resolution of the measuring instrument due to the multiscale nature of rough surfaces. Surface topography is a nonstationary random process and the length of a sample can influence the height distribution. Statistical parameters for the same rough surface can have different values when it is measured with instruments that have different resolution and scan length. Therefore, contact models based on these parameters may not be unique for a pair of rough surfaces due its scale-dependence.

One of the properties of rough surfaces is that when the surface is repeatedly magnified, more details of roughness are shown down to nanoscales. Additionally, roughness in all scales appears similar and can be characterized by fractal geometry. Fractal characterization of surface roughness is scale-independent and can provide information about the rough surface at all length scales.

Figure 1: A rough surface profile magnified repeatedly

The rough surface profile is continuous, nondifferentiable and self-affine. The nondifferentiability is caused by the fact that a tangent line or plane cannot be drawn at any point since more roughness details appears. It was found that the

Weierstrass-Mandelbrot (W-M) function satisfies all these properties and can be used to model the rough surface profile.

2. Normal contact stiffness

The contact of two rough surfaces can be modelled as a single rough surface in contact with a smooth surface. It can be represented as a single asperity in contact with a rigid flat plane and extended to many asperities.

2.1. Fractal elastic normal contact model

Normal contact stiffness of a single summit was described in the following form [4]:

\[
k_n = \frac{4}{3} E \left( \frac{3-D}{2-D} \right) R^{(3/2)} \omega^{1/2}
\]

where: \( E \) is equivalent Young’s modulus, \( R \) is asperity radius, \( D \) describes fractal parameter and \( \omega \) is an approach as shown in Fig. 2.

In [4] the total normal stiffness was obtained by integrating (1),

\[
K_n = \frac{4}{3} E \left( \frac{3-D}{2-D} \right) A_0 D_p \sigma_n^{1/2} \int_0^\infty \int_0^\infty \rho_{peak}(\zeta,t) d\zeta dt
\]

where: \( D_p \) is the density of peaks, \( A_0 \) is the nominal area, \( \sigma_n \) and \( \sigma \) are standard deviation and curvature of the rough surface, \( \rho_{peak} \) is probability density function of the normalized heights \( \zeta \) and curvatures \( t \).
Table 1: Surface roughness parameters for different machining processes [4]

<table>
<thead>
<tr>
<th>Machining Process</th>
<th>σ = Rq [µm]</th>
<th>R [µm]</th>
<th>D [1/mm²]</th>
<th>Rp [µm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fine sandblasted</td>
<td>0.832</td>
<td>40</td>
<td>500</td>
<td>5.3</td>
</tr>
<tr>
<td>Coarse sandblasted</td>
<td>5.13</td>
<td>30</td>
<td>230</td>
<td>15.3</td>
</tr>
<tr>
<td>Electrical discharge</td>
<td>8.94</td>
<td>19</td>
<td>160</td>
<td>48</td>
</tr>
</tbody>
</table>

2.2. FEA normal contact model

The contact stiffness of a single asperity can be also obtained in another way. Single asperities of various radius and height (Tab. 1) are modelled in Abaqus/CAE finite element analysis system. Roughness parameters were taken from the experiment described in [4]. 4-node linear tetrahedron elements (C3D4) were used to model three dimensional models of half of the asperities. 3-node 3-D rigid triangular facet (R3D3) elements were used to model a rigid flat plane.

The total contact stiffness can be determined by integrating the microstiffnesses from Eqn (1) or from the FE analysis above using the joint probability density function as shown in [2] or [4].

3. A numerical example

Contact stiffness is used as a subroutine UINTER in the Abaqus finite element analysis system to model a shrink-fit joint (Fig. 4). The shrink-fit joint was analyzed in the Abaqus as an axisymmetric model (Fig. 5).

4. The results

Contact pressures along the interface for different machining processes are shown in Fig. 6.

References


Dynamics of bodies under symmetric and asymmetric orthotropic friction forces

Olga A. Silantyeva¹, Nikita N. Dmitriev²

¹,²Faculty of Mathematics and Mechanics, Saint-Petersburg State University
198504, Universitetski pr.28., Peterhof, Saint-Petersburg, Russia
e-mail: olga.silantyeva@gmail.com ¹, e-mail: dn7@rambler.ru ²

Abstract

Recent studies show that the anisotropic behavior of frictional forces is an important factor in contact problems. The research provides a detailed explanation of dynamical behavior of a material point and bodies with circular and elliptical contact areas on the rough surface. Symmetric and asymmetric orthotropic frictional forces were mainly studied. An analytical model is presented for a material point. The results for a circular contact area are provided. Some situations are analyzed numerically. An example illustrates the influence of anisotropic effects on the evolution of dynamical characteristics of a mass point.

Keywords: anisotropic friction, asymmetric friction, orthotropic friction, elliptical contact area, circular contact area, mass point dynamic

1. Introduction

Anisotropy of friction forces appears in many cases. Wear, special surface textures, plasticity, changes in structure of surface layer, all lead to a situation when friction coefficients depend on the direction of sliding. Experimental and theoretical investigations of the phenomenon are done in the papers [1, 2, 3] and others. The effect of frictional anisotropy influences significantly the dynamical characteristics of the body motion on the rough surface.

The objective of the study is to provide a complete theory to analyze anisotropic effects in different cases. We point out main features dedicated to orthotropic symmetric and orthotropic asymmetric friction.

Investigations were done on a circular area with orthotropic friction and regular pressure distribution in [7], an elliptical area with orthotropic friction, regular and linear pressure distribution in [6], a mass point with asymmetric friction in [4] and a ring with asymmetric friction in [5].

In the paper a short description is presented, regarding the developed theory for a mass point case with asymmetric friction force.

2. Problem statement

Let us select the coordinate system thus, that friction coefficients in the positive directions of axes are \( f_{x+} \), \( f_{y+} \) and in negative directions are \( f_{x-} \), \( f_{y-} \) and \( f_{x+} \geq f_{x-} \), \( f_{y+} \geq f_{y-} \).

Velocity of a mass point is:
\[
\mathbf{v} = \dot{x} \mathbf{i} + \dot{y} \mathbf{j} = v (\cos \vartheta \mathbf{i} + \sin \vartheta \mathbf{j}),
\]
where \( \dot{x}, \dot{y} \) – projections of a velocity vector \( \mathbf{v} \), whose value is \( v \) and direction is \( \vartheta \).

Thus, the matrix of friction in the selected coordinate system is:
\[
Q = \begin{pmatrix}
    f_{x0} + f_{x1} \text{sign} (\dot{x}) & 0 \\
    0 & f_{y0} + f_{y1} \text{sign} (\dot{y})
\end{pmatrix},
\]
with
\[
f_{x0} = \frac{f_{x+} + f_{x-}}{2}, \quad f_{y0} = \frac{f_{y+} + f_{y-}}{2}
\]
and
\[
f_{x1} = \frac{f_{x+} - f_{x-}}{2}, \quad f_{y1} = \frac{f_{y+} - f_{y-}}{2}.
\]

A hodograph of friction force in that case is non-central symmetric (see [4]).

Projections of friction forces applied to the material point are as follows:
\[
T_x = -mg (f_{x0} + f_{x1} \text{sign} (\dot{x})) \cos \vartheta = -mg (f_{x0} + f_{x1} \text{sign} (\cos \vartheta)) \cos \vartheta,
\]
\[
T_y = -mg (f_{y0} + f_{y1} \text{sign} (\dot{y})) \sin \vartheta = -mg (f_{y0} + f_{y1} \text{sign} (\sin \vartheta)) \sin \vartheta.
\]

Equations of motion for the mass point on normal and tangential axes of Frenet-Serret frame are the following:
\[
m \ddot{v} \cos \vartheta + T_x \sin \vartheta,
\]
\[
m \ddot{v} \sin \vartheta + T_y \cos \vartheta.
\]

3. Selected results

Figure 1 shows trajectories of a movement of a material point under asymmetric orthotropic friction (with \( f_{x+} = 0.42, \ f_{y+} = 0.6, \ f_{x-} = 0.21, \ f_{y-} = 0.3 \) and symmetric orthotropic friction (with \( f_s = 0.42, \ f_s = 0.6 \)). Each trajectory corresponds to initial velocity \( 1 \) m/s directed to the bisector of a relative quadrant.

In the case of symmetric orthotropic friction (Fig.1b) trajectories are smooth curves. Tangents of these curves at the stopping point are parallel to axes \( Ox (f_x > f_y) \). A velocity vector during sliding rotates to the direction along \( Ox \) axes.

In the case of asymmetric friction (Fig.1a) the situation is different. It may occur, that in one quadrant friction coefficients have the following relation \( f_x > f_y \) (quadrants I, II, III) and in other quadrant \( f_y < f_x \) (quadrant IV). In the first case tangents to the trajectories at the final point are parallel to \( Ox \) axis. In the second case tangents are parallel to \( Oy \) axis.

Thus, with asymmetric friction we take into account directions of velocities in each quadrant.
In the work [5] some results regarding circular contact area with asymmetric friction are presented. In the case of zero initial linear velocity of the mass center and non-zero initial angular velocity, mass center gets acceleration directed to the third quadrant, if coefficients of friction along axes are two times higher than versus axes.

In the case of non-zero initial linear velocity and non-zero initial angular velocity, the velocity vector at the stopping point will be directed to the quadrant with minimal values of friction coefficients (to the third quadrant for the case described here). However, if the initial motion is translational, it stays translational up to the final point.

4. Conclusion

The methods are sequentially developed to consider effects of anisotropy of friction forces for a number of cases. The research tends to define the exact way of calculating friction forces and the movement of predicting bodies. This may be useful to better understand the basis of the process. Furthermore, the more information we get about frictional behavior, the more carefully wear and fatigue in contact pair are predicted.

The main results are:

- Differential equations describing dynamical behavior of material point, circular plate, elliptical plate in case of frictional anisotropy are developed.
- Some numerical results regarding symmetric orthotropic and asymmetric orthotropic friction are presented.
- In the case of symmetric orthotropic friction
  - For both circular and elliptic plates it is postulated that spinning and sliding finish simultaneously.
  - For elliptic and circular contact areas sliding regimes depend on the interrelations between inertia moment and frictional coefficients.
  - Initial orientation of elliptic area also acts upon the dynamical behavior.
- In the case of asymmetric orthotropic friction
  - The contact area should be partitioned due to relations between friction coefficients.
  - Direction of the velocity vector at the stopping point result from interrelations between frictional coefficients.

References

History-dependent variational inequalities in contact mechanics

Mircea Sofonea
Laboratoire de Mathématiques et Physique, Université de Perpignan Via Domitia
52 Avenue Paul Alday, 66 860 Perpignan, France
e-mail: sofonea@univ-perp.fr

Abstract

We present a new abstract existence and uniqueness result in the study of history-dependent variational inequalities. The proof is based on arguments of monotonicity, convexity and a fixed point result. Such kind of inequalities arise in the study of various models of quasistatic contact. In order to provide an example we consider a mathematical model which describes the frictional contact between a viscoelastic body and an obstacle, the so-called foundation. The contact is described with normal compliance and is associated to a slip-rate dependent version of Coulomb’s law of dry friction. We prove that the model casts in the abstract setting of history-dependent variational inequalities, with a convenient choice of spaces and operators. Further, we apply the abstract result to obtain a unique weak solvability of the contact model.

Keywords: history-dependent variational inequality, viscoelastic material, frictional contact, normal compliance, weak solution.

1. Introduction

The theory of variational inequalities plays an important role in the study of both the qualitative and numerical analysis of nonlinear boundary value problems arising in Mechanics, Physics and Engineering Science. For this reason the mathematical literature dedicated to this field is extensive and the progress made in the last fours decades is impressive. A part of this progress was motivated by new models arising in Contact Mechanics. The heart of this theory is the intrinsic inclusion of free boundaries in an elegant mathematical formulation. References in the field are [1, 4, 7], among others. Applications of variational inequalities in Mechanics and Engineering Sciences and, more specifically in Contact Mechanics, can be found in [2, 3, 5, 6, 8, 9, 10] and the references therein.

The aim of the paper is to prove a new existence and uniqueness result in the study of history-dependent variational inequalities and to apply it in the analysis of several quasistatic contact problems. The class of quasivariational inequalities considered represents a general framework in which a large number of quasistatic contact problems can be cast. Therefore, the paper provides arguments and tools which can be useful to prove the unique solvability of a large number of quasistatic contact problems. The intention of the paper is to illustrate the cross fertilization between modelling and applications, on one hand, with nonlinear analysis, on the other hand. Thus, within the particular setting of quasistatic process, it is shown how the models in Contact Mechanics lead to new types of variational inequalities and, conversely, how the abstract results on variational inequalities can be applied to prove the unique solvability of the corresponding contact problems.

2. History-dependent variational inequalities

Let $X$ be a real Hilbert space, $Y$ a normed space, $C([0, +\infty); X)$ and $C([0, +\infty); Y)$ the spaces of continuous functions defined on $\mathbb{R}_+$ with values in $X$ and $Y$, respectively. Given a subset $K \subset X$, $C([0, +\infty); K)$ denotes the set of continuous functions defined on $\mathbb{R}_+$ with values in $K$. Consider the operators $A : \mathbb{R}_+ \times K \to \mathbb{R}$ and $f : [0, +\infty) \to \mathbb{R}$. Based on this notation the following problem arises: find a function $u \in C([0, +\infty); K)$ such that

\begin{equation}
(Au(t), v - u(t)) + j((Su(t), u(t), v) = 0 \quad \forall v \in K, \quad \forall t \in \mathbb{R}_+, \tag{1}
\end{equation}

where $j$ is a convex and Lipschitz continuous functional.

Following the terminology introduced in [10] and used in various papers, condition (6) shows that the operator $S$ is a history-dependent operator. Such kind of operators arise both in functional analysis, theory of differential and partial differential equa-
tions, and in Contact Mechanics. Some simple examples in functional analysis are the integral operator and the Volterra-type operators. In Contact Mechanics, history-dependent operators could arise both in the constitutive law of the material and in frictional contact conditions. The memory term in the viscoelastic constitutive laws, the total slip, the total slip-rate and the accumulated penetration represent simple examples of history-dependent operators.

The main abstract result stated and proved in this paper is:

**Theorem 1.** Assume that (2)–(7) hold. Then, the variational inequality (1) has a unique solution \( u \in C(\mathbb{R}^+; K) \).

The proof of Theorem 1 is based on monotonicity, lower semicontinuity and fixed point arguments.

### 3. A slip-rate frictional contact problem

A large number of quasistatic contact problems with elastic, viscoelastic and viscoplastic materials lead to a variational inequality of the form (1) in which the unknown is either velocity or displacement field. In both cases the abstract result provided by Theorem 1 works. We provide below such an example in which the main variable is the velocity field.

The physical setting is presented in Figure 1 and is described below. A viscoelastic body occupies a regular domain \( \Omega \subset \mathbb{R}^d \) with a surface \( \Gamma \) partitioned into three disjoint measurable parts \( \Gamma_1, \Gamma_2 \) and \( \Gamma_3 \), such that \( \text{meas}(\Gamma_1) > 0 \). The body is clamped on \( \Gamma_1 \), so the displacement field vanishes there. Surface tractions of density \( f_s \) act on \( \Gamma_2 \) and volume forces of density \( f_p \) act in \( \Omega \). Moreover, the body is or can arrive in contact with an obstacle on \( \Gamma_3 \), the so-called foundation. We are interested in the evolution process of the mechanical state of the body in the unbounded interval of time \( \mathbb{R}^+ = [0, +\infty) \) and we assume that the forces and tractions change slowly in time so that the acceleration of the system is negligible.

Thus, the classical formulation of the contact problem we consider is the following: find a displacement \( u : \Omega \times \mathbb{R}^+ \to \mathbb{R}^d \) and a stress field \( \sigma : \Omega \times \mathbb{R}^+ \to \mathbb{S}^d \) such that, for all \( t > 0 \),

\[
\sigma(t) = A(e(u(t))) + g
\]

\[+ \int_0^t \mathcal{L}(t - s) e(u(t)) ds \quad \text{in} \quad \Omega,
\]

\[\text{Div } \sigma(t) + f_s(t) = 0 \quad \text{in} \quad \Omega,\]

\[u(t) = 0 \quad \text{on} \quad \Gamma_1, \quad \sigma(t) \nu = f_p(t) \quad \text{on} \quad \Gamma_2, \quad -\sigma(t) \nu = p(u(t) - g) \quad \text{on} \quad \Gamma_3.\]

\[
\|\sigma(t)\| \leq \mu(\|u(t)\|)\|\nu(t)\|,
\]

\[-\sigma(t) \nu = p\|u(t)\|\|\nu(t)\| \quad \text{on} \quad \Gamma_3.\]

\[
\|u(t)\| = u_0 \quad \text{in} \quad \Omega. \quad (14)
\]

Equation (8) represents the viscoelastic constitutive law in which \( A \) is a nonlinear operator which describes the properties of the material, \( \mathcal{L} \) is a nonlinear operator which describes its elastic behavior and \( \mathcal{C} \) represents the relaxation tensor. Equation (9) represents the equilibrium equation where \( \text{Div } \sigma \) represents the divergence operator, i.e. \( \text{Div } \sigma = (\sigma_{i,j}) \). Conditions (10) and (11) are the displacement and traction boundary conditions, respectively. Conditions (12) and (13) represent the contact condition with normal compliance and the Coulomb law of dry friction, respectively, where \( u_n \) is the normal displacement, \( \sigma_n \) denotes the normal stress, \( \sigma_t \) represents the tangential traction and \( u_t \) is the tangential part of the velocity field, called also the slip-rate. Moreover, \( g \) is the gap function, \( p \) represents the normal compliance functions and \( \mu \) denotes the coefficient of friction. Note that in (13) we assume that the coefficient of friction depends on the slip-rate. This dependence makes the considered contact model more accurate. Nevertheless, it gives rise to additional difficulties in its analysis. Finally, (14) represents the initial condition in which the function \( u_0 \) denotes the initial displacement field.

![Figure 1: Physical setting](image)

In the study of a problem (8)–(14), under appropriate assumptions on the data, we show that the velocity field \( w = u \) satisfies a history-dependent variational inequalities of the form (1). The abstract result provided by Theorem 1 is used to prove the existence of a unique weak solution to a given contact model.

### References


Development of friction constitutive relations for polymers

Alfred Zmitrowicz
Institute of Fluid-Flow Machinery, Polish Academy of Sciences
J. Fiszera 14, 80-231 Gdańsk, Poland
e-mail: azmit@imp.gda.pl

Abstract

Friction models of polymers must be considered in terms of different scales and different friction processes at external boundaries of solids and inside the materials. In the contribution the author proposes friction relations for polymers taking into account friction anisotropy and heterogeneity, adhesion and hysteresis components, various models of polymeric macromolecules and different modes of their kinematics.

Keywords: friction at solid’s boundaries, friction inside materials, friction anisotropy and heterogeneity, thermodynamic restrictions

1. Introduction

Complex materials require specific constitutive relations of friction. In certain polymers an intrinsic structure of solids acts upon friction, a microstructure evolution at a solid boundary induces variations of friction, see Ref. [1, 3]. Friction with anisotropy and heterogeneity effects at external boundaries of solids takes place in polymeric composites, semi-crystalline polymers, elastomers, self-lubricating polymers (see Fig. 1). Friction with anisotropy effects inside the material occurs in polymeric solutions, melts, liquid crystalline materials.

2. Equations of friction at boundaries of solid polymers

The dry friction linear equation with respect to the unit vector \( \mathbf{v} \) of the sliding velocity (see Tab. 1) is given by

\[
P_i = - p_n | \mathbf{C}_1 \mathbf{v} = \begin{array}{c} \hat{\mathbf{u}}_i \\ \mathbf{u}_i \end{array} \end{array} | \mathbf{C}_1 : R^2 \rightarrow R^2
\]

where, \( p_n \) is the friction force, \( p_n \) is the normal pressure, \( \mathbf{u}_i \) is the sliding velocity, \( \mathbf{C}_1 \) is the dry friction second-order tensor with simple cases of friction anisotropy included, \( R \) is the space of real numbers. The non-linear equation with respect to the unit vector \( \mathbf{v} \) given as a polynomial has the following form

\[
P_i = - p_n | \left[ \mathbf{C}_1 \mathbf{v} + \mathbf{C}_2 (\mathbf{v} \otimes \mathbf{v}) + \ldots \right]
\]

where, \( \mathbf{C}_2 \) is the fourth-order friction tensor with complex cases of friction anisotropy included, see Ref. [4]. Relation the friction tensors to oriented angle is described as follows

\[
\mathbf{C}_i = \mathbf{C}_i(\alpha_v), \quad \alpha_v \in \langle 0, 2\pi \rangle \quad i = 1, 2, \ldots
\]

where, \( \alpha_v \) is a measure of the oriented angle between the sliding direction \( \mathbf{v} \) and a reference direction (e.g. Ox-axis), due to the friction asymmetry effects are included. The friction equation can be defined as the sum of two single-term polynomials with respect to the unit vectors \( \mathbf{v} \) and \( \mathbf{n} \)

\[
P_i = - p_n | \left[ \mathbf{C}_1 \mathbf{v} + \mathbf{E}_1 \mathbf{n} \right], \quad \mathbf{v} \cdot \mathbf{n} = 0, \quad | \mathbf{n} | = 1
\]

where, \( \mathbf{n} \) is the unit vector normal to the sliding trajectory, \( \rho \) is the sliding trajectory curvature radius, \( \mathbf{E}_1 \) is the second-order tensor with friction heterogeneity effects included Ref. [3]. The constitutive relations are restricted by thermodynamics Ref. [5]:

\[
p_i(p_n, Rv, R_n^1) = Rp_i(p_n, \mathbf{v}, \mathbf{n}) \forall R
\]

(5)

\[
R^{-1} = R^T, \quad det R = \pm 1
\]

(6)

(b) the friction force power is always non-positive in every case

\[
P = \mathbf{p}_t \cdot \mathbf{u}_t \leq 0 \quad \forall \mathbf{u}_t
\]

(7)

where, \( R \) is the orthogonal tensor, \( P \) is the friction force power.

Figure 1: Boundary conditions of the contacting solid body

3. Two-component equations of friction for elastomers

At the external boundary of rubber the energy dissipation and friction are influenced by: (a) contact conditions (adhesion), (b) bulk properties (hysteresis effects). If two processes operate independently, the friction force can be expressed as the sum of two terms

\[
P_t = p_{adh} + p_{hyst}
\]

(8)

where, \( p_{adh} \), represents friction due to adhesion, \( p_{hyst} \) represents friction due to hysteresis. Adhesion occurs when a normal tensile force yields from contact of separate surfaces and is defined by Frémond law

\[
p_{adh} = \beta | p_n |, \quad p_{adh} \geq 0, \quad \beta \in \langle 0, 1 \rangle
\]

(9)
where, $\beta$ is the adhesive intensity, $p_n$ is the initially applied compressive force (the normal pressure). The adhesion law is valid in normal and tangential directions. A hysteresis loop (i.e., hysteresis forces versus displacements) defines the dissipated energy during bulk deformations in the near external boundary region.

4. Continuum-based equations of friction inside polymers

Polymeric materials exhibit a great number of macromolecules, in some cases considered as liquid-like continua. The equation of viscous friction of macromolecules (see Tab. 1) is given by

$$ p = -B \dot{u} , \quad B : R^3 \rightarrow R^3 $$  \hspace{1cm} (10)

$$ p = p_n + p_t , \quad \dot{u} = \dot{u}_n + \dot{u}_t $$  \hspace{1cm} (11)

where, in accordance with the principle of objectivity $\dot{u}$ is the relative velocity at the given point of the continuum (i.e., the macromolecules move relative to one another), $B$ is the viscous friction second-order tensor with friction anisotropy effects included. The polymer chains move with the highly anisotropic friction.

5. Microscopic-level models of friction inside polymers

In the frame of micromechanics, the macromolecules are independent kinematical elements. They idealize simple shapes and individual modes of motion. Their kinematics consists of translations, rotations and spinning. Two classical models of macromolecules exist. (a) Rouse model (1953), where the macromolecules are chains of spherical beads and springs (or rods), see Fig. 2. (b) De Gennes tube model (1971): with long macromolecules moving like a snake inside a long and narrow tube Ref. [6]. Polymers in liquid crystal phase can form long rigid molecules similar to rods and rigid flat molecules similar to discs Ref. [6]. Microscopic-level friction models can be applied in homogenization of the frictional behavior inside the polymeric materials and in Brownian dynamics of polymers with friction.

Figure 2: The Rouse model of the polymeric macromolecule, the chain of beads connected by springs (1953)

5.1. Friction of beads in the Rouse model

In the Rouse model, during motion the beads are affected by the surrounding molecules and friction forces arise e.g., viscous friction forces (located at bead centers since the beads have no volume). The springs simulate elastic properties of macromolecules.

5.2. Friction of rod-like and disc-like macromolecules

The rod-like and disc-like macromolecules may slide one against another. Energy dissipation occurs due to interfacial friction between these objects. Dry friction models are applied in this study. Two particular cases are considered: (a) friction of rod-like molecules under sliding and rolling, (b) friction of disc-like molecules under sliding and spinning. In both cases friction couples are present. In accordance with thermodynamics, power of a friction force couple is non-positive

$$ P = c \cdot \omega \leq 0 \quad \forall \omega $$  \hspace{1cm} (12)

where, $c$ is the friction force couple, $\omega$ is the velocity of rotation (or spinning). In Ref. [1] rod-like material microstructures are modelled as a Cosserat continuum. Dynamics of the rigid disc in the presence of anisotropic friction in the sliding plane is described in Ref. [2].

6. Conclusions

Relations for friction at the external boundaries of polymeric solids can be used to predict unlubricated contact in a number of cases, for dry sliding bearings, sliding seals, transmission belts, brakes, clutches, car tyres, prostheses of human joints, etc. Equations of friction inside the polymeric materials are practically applied in the simulation of various production processes in chemical industry.

References


Experimental Mechanics and Thermomechanics of Materials Related to Phase Transformation

organized by H. Tobushi and E. Pieczyska
Experimental investigation of the innovative flow control blowing devices

Andrzej Krzysiak
Institute of Aviation
Al. Krakowska 110/114, 02-256 Warszaw, Poland
e-mail: andkrzys@ilot.edu.pl

Abstract

The research of the use of new flow control methods and their implementation on real objects, especially on the aircrafts, is one of the priority directions of new technologies development. In the case of aircraft, the interest in new flow control methods is associated with potential opportunities to improve their aerodynamics and to reduce operating costs. This paper presents the results of experimental tests related to the application of the innovative flow control methods on aircraft lift surfaces and helicopter blades through the use of air blowing. Three different applications of the flow control blowing devices are presented. The experiments were carried out of the Institute of Aviation in low-speed wind tunnels.

Keywords: fluid mechanics, flow control, blowing devices

1. Introduction

Active flow control became widely used in many fields of science and technology and continues to be the subject of intensive experimental and numerical studies in a number of research centers. Improvement of the efficiency of currently used aircraft control systems or replace them by unconventional flow control methods, can be a source of measurable benefits. These benefits can still be significantly enhanced by the use of flow control operating in the Close Loop Control (CLC) System.

The literature describes a number of different flow control methods [1,2]. One of the methods of active flow control is an additional blowing on a wetted surface. In this method, properly targeted additional air jets increase energy of the flow. Boundary layer supplied with additional energy becomes less susceptible to separation, even at angles of attack higher than the critical (for the condition without blowing). A postponed flow separation contributed to the increase of maximum lift and simultaneously to drag decrease. This in turn, can improve the airplane aerodynamic performance.

In the paper three different applications of the flow control blowing devices on the aircraft wings and helicopter blades are shown. The presented blowing devices were used not only to the delay the flow separation, but also to control the load on the aircraft wing. Experimental studies relate to:

- flow control using conventional and self-supplying air jet vortex generators installed on the segment of the airfoil,
- separation control on the wing flap controlled by close loop system,
- wing load control using blowing fluidic devices.

2. Wind tunnels

The experiments were carried out in Institute of Aviation low speed wind tunnels T-1, T-3 and trisonic wind tunnel N-3. The low speed wind tunnel T-1 is a closed-circuit, continuous-flow wind tunnel with 1.5 meter diameter open test section. The range of freestream velocity is 15÷40 m/s. The T-3 low speed wind tunnel is an atmospheric, closed-circuit tunnel with an open test section of 5 meter diameter, which can reach velocity of 57 m/s. The N-3 wind tunnel is a blow-down type with partial re-circulation of the flow. It can operate in subsonic, transonic and supersonic flow regimes at Mach numbers M = 0.2÷1.2, 1.5 and 2.3. The closed test section have perforated top and bottom walls for tests at subsonic and transonic flow velocities and solid walls - at supersonic.

3. Experimental test results

3.1. Flow control using air jet vortex generators

Experimental studies of flow control using air jet vortex generators (both conventional and proposed self-supplying) were conducted in the wind tunnels T-1 and N-3. Air jet vortex generators, which are an alternative to traditional vane vortex generators consist of a number of small streams outgoing from the wetted surface and properly oriented with respect to the undisturbed flow. The interaction between the air jets and undisturbed flow generates a well-organized vortex structures, see Fig. 1.

![Figure 1: Air jet vortex generators](image)

Vortices are able to withstand the adverse pressure gradient appearing on the upper surface of the airfoil, at higher angles of attack. As a result the flow separation is delayed. The proposed self-supplying air jet vortex generators in comparison with conventional ones use the airfoil overpressure regions, as a source of the air for them.
The optimal design parameters of air jet vortex generators were obtained.

Experimental studies showed, that the proposed self-supplying air jet vortex generators produce such strong turbulence areas that their effectiveness is only slightly lower than optimal designed conventional ones and they can become an alternative to the previously used vortex generators. An important advantage of the use of self-supplying air jet vortex generators, instead of conventional ones, would be a significant simplification of their design, as they would not need to use an external air supply source (i.e. compressor), but would be supplied with air from the overpressure areas.

3.2. Separation control on the wing flap controlled by close loop system

Wind tunnel tests of flow control using an additional blowing on the airfoil segment equipped with the movable flap were performed. Blowing was performed through the set of nozzles located on the trailing edge of the main body of the airfoil. Air flow through the nozzles was controlled by a set of the electromagnetic valves located inside the model.

Minimization of air flow rate necessary to maintain the desired state of the boundary layer on the upper flap surface, required the implementation of such process in control loop. The main task of the close loop control (CLC) system used for fluidic active flow control was to keep flow attached, during flap movement, in the range from $\delta=0^\circ$ to $\delta=45^\circ$. This process was based on “on-line” analysis of the boundary layer. Pressure, measured by the sensors mounted on the upper flap surface (close to its trailing edge), created a feedback signal for a CLC system. This signal was analyzed by control unit. In the case of attached flow the coefficient $C_p$ of measured pressure had had a positive value and the valves were closed. In the case of a detached flow, the coefficient $C_p$ of measured pressure had had a negative value and the valves were opened. In Fig. 2, general idea of closed loop control system for fluidic active flow control is presented.

The work was performed under the European project “ESTERA”.

A new concept of active flow control system based on proposed blowing devices for the control of the aerodynamic load on aircraft wing was designed in Institute of Aviation. Two systems of fluidic control devices were tested. The first system was based on the nozzles blowing air in normal and inclined directions with respect to the upper wing surface. The nozzles were located at 40-70% of wing chord, as classical spoilers. The second system was based on specially designed nozzles located on a modified trailing edge surface. The fluidic control devices were supplied with air from the high pressure area situated at lower wing surface close to its leading edge or from compressor.

The experimental tests were performed in IoA low speed wind tunnel T-3. For these tests the model of semi-span wing (2.4 m span) situated vertically on the endplate in wind tunnel test section was used. The model was situated on two aerodynamic wall balances. To measure the load distribution along the semi-span wing model the 8 strain-gauge bridges were glued to the model front spar. Wind tunnel test were performed at Mach number $M = 0.1$.

The study was carried out in the framework of the European project STARLET.

Figure 3: The semi-span wing in the wind tunnel T-3

4. Conclusions

- Three different applications of the flow control blowing devices on the aircraft wings and helicopter blades were presented.
- The proposed innovative flow control blowing devices may be an alternative for conventional mechanical control systems.
- Effectiveness of the tested blowing devices is comparable with conventional control systems.

References

Evaluation of the properties of polymeric foams with shape memory under load

Dominik Kukla\textsuperscript{1}, Maria Staszcak\textsuperscript{2}, Elżbieta Pieczyska\textsuperscript{3}, Marcin Heljak\textsuperscript{4}, Karol Szłazak\textsuperscript{5}, Wojciech Świątkowski\textsuperscript{2}, Mariana Cristea\textsuperscript{7}, Hisaaki Tobushi\textsuperscript{8}, Shunichi Hayashi\textsuperscript{9}\textsuperscript{*}

\textsuperscript{1,2,3} Institute of Fundamental Technological Research
Pawiński 5b, 02-106 Warsaw, Poland
e-mail: dkukla@ippt.pan.pl, mstasz@ippt.pan.pl, epiecz@ippt.pan.pl
\textsuperscript{4,5,6} Faculty of Materials Science Warsaw University of Technology
Woloska 141, 02-507 Warsaw, Poland
e-mail: mheljak@inmat.pw.edu.pl, kszlazak@inmat.pw.edu.pl, wswieszk@inmat.pw.edu.pl
\textsuperscript{7} Petru Poni Institute of Macromolecular Chemistry
Aleea Grigore Ghica Voda 41A, 700487 Iasi, Romania
e-mail: mcristea@icmpp.ro
\textsuperscript{8} Aichi Institute of Technology
1247 Yachigusa, Yagusa-cho, Toyota City, Aichi Prefecture 470-0392, Japan
e-mail: tobushi@aitech.ac.jp
\textsuperscript{9} SMP Technologies Inc.
Ebisu First Place 22-8, Ebisu, Shibuya-Ku, 150-0013 Tokyo, Japan
e-mail: hayashi@smptechno.com

Abstract

The paper presents the results of experimental investigation on polymer foam with shape memory properties. The research is focused on characterization of the microstructure of the foam and understanding the mechanisms of deformation under static and dynamic loading. Up till now, selected experimental techniques have been applied. Dynamic Mechanical Analysis (DMA) allows determining the extent of the value of the glass transition temperature under different load conditions, which also reveals the transformation temperature range for the SMP foam. Scanning electron microscopy (SEM) shows the foam microstructure in various scales, while X-ray tomography gave 3D microstructure results presenting in addition mechanism of the cells deformation and changes in their geometry under 30\% and 50\% strain. BOSE system enables obtaining the results on dynamic loading.

Keywords: shape memory polymer foam, Dynamic mechanical analysis, Glass transition temperature, X-ray tomography

1. Introduction

Shape memory polymers (SMP), like some metal alloys exhibit the shape memory effect and belong to a group of smart materials. The functional properties of the SMP exist due to the difference between characteristics of molecular motion above and below the glass transition temperature \( T_g \) \cite{1}. If SMP is deformed at temperatures above \( T_g \) and cooled down to temperatures below \( T_g \) by holding the deformed shape constant, the shape is fixed and the SMP can carry larger load. If the shape-fixed SMP element is heated up to temperatures above \( T_g \) under no load, the original shape is recovered \cite{2}.

Among the SMPs, the polyurethane shape memory polymer (PU-SMP) has been often practically used, since the rigidity of polyurethane is high in comparison to other polymers \cite{3}. Especially, the polyurethane shape memory foam is recently drawing attention, since it has not only the shape memory material characteristics, but also the particular foam structure \cite{3}. The SMP foam is characterised by impact relaxation, energy absorption and heat insulating properties. Therefore, the SMP foam is applied in aerospace and aircraft industry, biomedical elements, pharmacy; e.g. drug delivery systems, as well as in textile and responsible packaging industry.

2. Methodology

In the study, in order to learn more about the new material, some experimental investigation on the SMP foam structure and mechanical properties have been carried out.

Dynamic Mechanical Analysis allowed determining the SMP foam region of the glass transition temperature \( T_g \).

The scanning electron microscopy illustrates of the skeletal foam structure.

X-ray tomography shows the foam structure in various micro scales, also under loading. To this end, a sample of foam was used in a state of static deformation at 30\% and 50\%. It allowed exploring the mechanisms of strain in the range of elastic foam. The SMP foam sample with size 10x10x10 mm was mounted in a holder of the Sky scanner Scan(\textsuperscript{\textregistered}) with the spatial resolution in the range of hundreds of nanometers. In terms of volume is equal to or better than the resolution synchrotron tomography. The device uses open source X-ray of the LaB6 cathode. The test was performed in room conditions; at temperature 22\(^\circ\)C and in humidity 45\%.

Moreover, a BOSE dynamic loading system was used in order to evaluate the foam strength and assess its suitability for characterization of the mechanical properties of the SMP foams.
3. Results

3.1. Scanning electron microscopy SEM

Figure 1: Microstructures of SMP foam by SEM (x 100)

Observations of the SMP foam by SEM have been conducted at magnifications x 100, x 200 and x 500. An example of the obtained microstructure with magnification x100 is shown in Figure 1. It can be noticed that the microstructure is characterized by a homogeneous distribution of regular cells with sizes in the range of approximately 300 μm - 500 μm.

3.2. Dynamic mechanical analysis DMA

Example of the results obtained by DMA for the SMP foam sample subjected to shear with frequency 1 Hz and ramp temperature increase 2 °C/min, is shown in Figure 2.

Figure 2: DMA results. Variation of storage modulus $G'$, loss modulus $G''$ and loss factor $\tan \delta$ with temperature $T$

Variation of the storage modulus ($G'$), loss modulus ($G''$) and loss factor ($\tan \delta$) with temperature enable to estimate glass transition temperature range, approximately from -10°C to -6°C.

3.3. X-ray tomography

X-ray tomography shows the SMP foam cells in 3D, moreover of varying geometry under static tensile load. Example of the results is shown in Fig. 3.

Figure 3: Reconstructed images with visible cells. By: longitudinal (a), transverse (b) and sagittal (c) intersection. White fiber - at deformed state ε=30%, yellow - at ε=50%

The obtained X-ray tomography results allowed for quantitative analysis of changes in equivalent diameters of the foam cells and expected ratio of the cells at the strain of 30% and 50%.

4. Conclusions

The obtained results show some properties of shape memory polymer foam – a new shape memory material with foam structure and high potential in applications.

Results of dynamic mechanical analysis allowed estimating the value of the SMP foam glass transition temperature; in the range of approximately from -10°C to -6°C.

Observations by SEM reveal that the SMP foam microstructure is characterized by a homogeneous distribution of regular cells with sizes in the range of 300 μm - 500 μm.

Analysis of the SMP foam spatial structure performed by X-ray tomography showed slight changes both in the cell geometry (shape impact) as well as in the cell size (equivalent diameter), estimated at 30% and 50% deformation. This may be caused by a negative Poisson's ratio of the foams investigated by the responsibility of their skeletal structure.

References


Gum Metal – unique properties and results of initial investigation of the new titanium alloy

Elżbieta Pieczyska1, Michał Maj2, Tadahiko Furuta3, Shigeru Kuramoto4

1,2 Department for Strength of Materials, Institute of Fundamental Technological Research, Polish Academy of Sciences
Pawińskiego 5 B, 02-106 Warsaw, Poland
e-mail: epiecz@ippt.pan.pl1, mimaj@ippt.pan.pl2

3 Toyota Central Research & Development Laboratories, Inc.
Nagakute, Aichi, 480-1192, Japan
e-mail: e0646@mosk.tytlabs.co.jp

4 Department of Mechanical Engineering, College of Engineering, Ibaraki University
4-12-1, Nakanarusawa, Hitachi, 316-8511
e-mail: kuramoto@mx.ibaraki.ac.jp

Abstract

Initial results of effects of thermomechanical couplings in the β-Ti alloy, Gum Metal, subjected to tension are presented. An MTS testing machine allows obtaining stress-strain curves with high accuracy, while fast and sensitive infrared camera allows estimating temperature changes of the sample during the deformation process. The obtained mechanical characteristics confirm an ultra-low elastic modulus and high strength of the Gum Metal. The infrared measurements enable to indicate average or maximum temperature change accompanying the alloy deformation process, to estimate thermoelastic effect, related to the yield point in solids, whereas the temperature distribution on the sample surface enables to investigate localisation effects, leading to the sample necking and rupture.

Keywords: Gum Metal, Titanium alloy, Elastic properties, Nonlinear elasticity, thermomechanical couplings, infrared camera

Introduction, results and discussion

The aim of the research is an investigation of the effects of thermomechanical couplings in new multifunctional titanium alloy, developed in Japan in the beginning of the 21st century, called Gum Metal, since it combines high elasticity and flexibility of rubber and strength of metal. It is a beta-type titanium alloy with a simple body-centered-cubic (bcc) crystal structure and composition fundamentally expressed as Ti3 (Ta, Nb, V)+(Zr, Hf, O); prepared by a powder sintering process.

Photograph of the experimental set-up designed for the Gum Metal tension test showing the MTS Testing Machine and Flir Co Phoenix Infrared System is presented in Fig. 1.

Figure 1: Photograph of the experimental set-up

The Gum Metal is characterized by an ultra-low elastic modulus with very high strength, superelastic nature - one digit higher in nonlinear elastic deformation (~2.5%) compared to other metallic materials, super-plastic nature allowing cold plastic working without hardening up to (~90%), very low linear coefficient of thermal expansion (similar to Invar) and a constant elastic modulus in the temperature range from -200 °C to +250 °C (similar to Einvar) [1,2,3,4]. A comparison of the mechanical curves obtained for Gum Metal and Steel during tension with strain rate 2x10^{-3}s^{-1} is presented in Fig. 2.

Figure 2: Comparison of stress vs. strain curves for Gum Metal and stainless steel - 316L up to the sample rupture

During the loading, the mechanical parameters and infrared radiation from the specimen surface were simultaneously recorded (Fig. 1). Stress-strain characteristics were found and the temperature changes of the specimens during the deformation process have been elaborated. The stress and the strain quantities were related to the current (instantaneous) values of the specimen cross-section values, obtaining so called true stress (σtrue) and true strain (εtrue) values, presented in the diagrams (Fig. 2, Fig. 4).

*The research has been carried out with the support of the Polish National Centre of Science under Grant No. 2014/13/B/ST8/04280. Authors are also grateful to Leszek Urbaniski and Maria Staszczak for obtaining mechanical data, elaborating diagrams and many useful experimental remarks.
The sample temperature distribution, called thermograms, are obtained in contactless manner by a fast and sensitive infrared camera; Fig. 1. An example of the temperature distributions on the Gum Metal specimen and the temperature profiles along the specimen center obtained for the tension test just before the specimen rupture are shown in Figure 3. A strong strain localization, characterized by a high temperature increase, was captured in the presented thermogram. Such thermograms and temperature profiles demonstrate a possibility to study a more advanced deformation process leading to the specimen necking and rupture. Moreover, a mean temperature during the deformation process has been calculated, using the infrared measurement methodology, elaborated in IPPT PAN for other materials [5]. The sample temperature referred to the deformation parameters allows indicating a limit of the alloy reversible deformation with high accuracy, according to the thermodynamic laws and Kelvin formula [6].

However, it can be noticed looking at Fig. 4 that a maximal drop in the Gum Metal specimen temperature occurs significantly earlier than the limit of the reversible deformation, macroscopically estimated. It means that so large limit of the reversible elastic deformation (nonlinear) stressed as the Gum Metal "super" properties [1,2,3,4], follows from other deformation mechanisms and probably can not be described by the Lord Kelvin formula. The result is quite different from those observed until now for other titanium alloys, stainless steel and alumina.

References


Shape memory polymer – shape fixity and recovery in cyclic loading

Maria Staszczak1, Elżbieta Pieczyska2, Michał Maj3, Katarzyna Kowalczyk-Gajewska4, Mariana Cristea5, Hisaaki Tobushi6, Shunichi Hayashi7

1,2,3 Institute of Fundamental Technological Research PAS
Pawinskiego 5b, 02-106 Warsaw, Poland
e-mail: mstasz@ippt.pan.pl, epiecz@ippt.pan.pl2, mimaj@ippt.pan.pl3, kgowalcz@ippt.pan.pl4,
5 "Petru Poni" Institute of Macromolecular Chemistry
Alea Grigore Ghica Voda 41A, 700487 Iasi, Romania
email: mcristea@icmpp.ro
6 Aichi Institute of Technology
1247 Yachigusa, Yakusa-cho, Toyota City, Aichi Prefecture 470-0392, Japan
e-mail: tobushi@aitech.ac.jp
7 SMP Technologies Inc.
Ebisu First Place 1-22-8, Ebisu, Shibuya-Ku, 150-0013 Tokyo, Japan
e-mail: hayashi@smptechno.com

Abstract

The paper concerns investigation of polyurethane shape memory polymer (SMP) properties. Shape fixity and shape recovery, important parameters for the SMP applications, were quantitatively estimated in thermomechanical cyclic loading; three subsequent thermomechanical loading cycles were performed. It was observed that the shape fixity is proper and does not depend on the cycle number. The obtained mean values of shape fixity parameters are 97-98 %. Although the shape recovery is poor (=83 %) in the first cycle of the thermomechanical loading, it is excellent in the subsequent cycles (=99-100 %). The evaluated parameters confirm good shape memory properties of the SMP.

Keywords: shape memory polyurethane, shape fixity, shape recovery, thermomechanical loading, cyclic loading

1. Introduction

Shape memory polymers (SMP) represent a class of smart materials which has competitive advantages compared to shape memory alloys: low weight, low cost, easy manufacturability, good shape fixity and recovery properties [1].

Shape memory polymers are able to respond to a particular external stimulus, such as heat, light, moisture, etc.; however, the thermo-responsive SMPs are the most common.

The functional properties of the thermally responsive polymers are related to the glass transition temperature $T_g$, in which the characteristics of the polymer behaviour are affected by molecular motion that varies depending on the temperature. If a SMP is deformed at temperature above $T_g$ and cooled down to temperature below $T_g$ by holding the deformed shape constant, the deformed shape is fixed. If the shape-fixed element is heated up again to temperature above $T_g$ under no-load conditions, it recovers its original shape and properties [2]. The SMP shape memory behaviour was investigated in the paper.

2. Materials and specimens

The material used in the performed experiment was the polyurethane shape memory polymer, characterised by $T_g$ approximately equal to 45°C and degree of crystallinity of ≈5%.

A dynamic mechanical analysis (DMA) was carried out in order to characterize the shape memory polyurethane properties. The DMA allowed to obtain important polymer parameters, like glass transition temperature ($T_g$), as well as storage modulus ($E'$), loss modulus ($E''$) and loss factor ($\tan \delta$), depending on temperature [3]. The DMA experiment was conducted during bending deformation, with frequency of force oscillation 1 Hz and heating rate 2 °C/min.

The results obtained in DMA suggest that the PU-SMP material fulfills some preliminary demands to perform as a shape memory polymer: a high glass elastic modulus $E'_g$ (1250 MPa), proper value of the rubber modulus $E''_r$ (12.1 MPa) and a high ratio of $E'_g/E''_r$ (103). The $T_g$ determined as $\tan \delta$ peak is equal to 45°C.

3. Experimental procedure

The SMP specimens have been subjected to thermomechanical cyclic loading program performed on a MTS 858 testing machine equipped with a thermal chamber.

The general description of the thermomechanical loading program is presented in Tab. 1, while Fig. 1 shows the stress-strain curves obtained during the test.

![Figure 1: SMP true stress $\sigma$ - true strain curves $\varepsilon$ during thermomechanical test with strain rate 10^{-3}s^{-1}. Number denotes process stage: 1 – loading at $T_g$+20°C, 2 – cooling down to $T_g$-20°C, 3 – unloading at $T_g$-20°C, 4 – heating to $T_g$+20°C](image-url)

* The research has been carried out with the support of the Polish National Center of Science under Grant No. 2011/01/M/ST8/07754
At first, the specimen was heated to high temperature $T_h = 65^\circ$C ($T_h + 20^\circ$C). Then, it was loaded at $T_h$ till maximum strain of 20% with strain rate of $10^{-3}$s$^{-1}$ (Fig. 1 – 1st stage). While maintaining the strain, the specimen was cooled down to $T = 25^\circ$C ($T_h - 20^\circ$C); (Fig. 1 – 2nd stage). After that, the specimen was unloaded at $T_l$ with the same strain rate (Fig. 1 – 3rd stage). During the subsequent heating from $T_l$ to $T_h$ under no-load conditions the SMP specimen almost recovered its original shape (Fig. 1 – 4th stage); however a residual strain $\varepsilon_r$ was recorded. The thermomechanical loading cycle was repeated three times.

### Table 1: Description of thermomechanical loading program

<table>
<thead>
<tr>
<th>Heating up to 65°C</th>
<th>Loading up to 20% at $T_h = 65^\circ$C</th>
<th>Cooling down to 25°C</th>
<th>Unloading to 0 N at $T_l = 25^\circ$C</th>
<th>Heating up to 65°C</th>
</tr>
</thead>
</table>

The following equations were used in order to define shape fixity $R_f$ (1) and shape recovery $R_r$ (2) parameters in the N-th cycle, respectively:

$$R_f = \frac{\varepsilon_{ir} - \varepsilon_{ir}(N)}{\varepsilon_{max}(N)} \times 100\% \quad (1)$$

$$R_r = \frac{\varepsilon_{ir} - \varepsilon_{ir}(N)}{\varepsilon_{ir}(N) - \varepsilon_{ir}(N-1)} \times 100\% \quad (2)$$

where $\varepsilon_{max}$ denotes maximum strain, $\varepsilon_{ir}$ – the strain obtained after unloading at $T_l$ and $\varepsilon_{ir}$ – irrecoverable strain, i.e. strain obtained after heating up to $T_h$ under no-load conditions.

The parameters of shape recovery $R_r$ and shape fixity $R_f$ have been estimated for 3 subsequent thermomechanical cycles. An example of the obtained results is shown in Tab. 2. As can be seen in the table, the shape fixity has not changed markedly in subsequent cycles, while the shape recovery is poor for the first cycle and excellent for the subsequent cycles of the thermomechanical loading.

### Table 2: Example of shape fixity and shape recovery parameters in three cycles obtained for SMP

<table>
<thead>
<tr>
<th>Cycle No</th>
<th>Parameters, %</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Shape fixity</td>
</tr>
<tr>
<td>1</td>
<td>98</td>
</tr>
<tr>
<td>2</td>
<td>98</td>
</tr>
<tr>
<td>3</td>
<td>97</td>
</tr>
</tbody>
</table>

### Figure 2: Experimental results obtained during three SMP thermomechanical loading cycles conducted with strain rate of $10^{-3}$s$^{-1}$: a) chamber temperature vs. time; b) true stress $\sigma$ vs. time; c) true strain $\varepsilon$ vs. time

Shape fixity shows the measure how a temporary deformed shape is fixed (stored), while shape recovery means the measure of how well the original permanent shape is recovered (or restored).

The cyclic thermomechanical analysis has been used to quantify the shape memory effects of the rigid polyurethane shape memory polymer ($T_h = 45^\circ$C).

Important parameters, crucial for the SMP applications, have been evaluated in three thermomechanical loading cycles. The obtained average value of the shape fixity is 97-98 %, while the shape recovery is 83 % in first cycle and 99-100 % in subsequent thermomechanical loading cycles.

Estimation of the application parameters, i.e. shape fixity and shape recovery, carried out at maximum strain of 20 % and at temperature range $T_l - 20^\circ$C; $T_h + 20^\circ$C, gave reasonable values and confirmed good shape memory properties of the polymer.

### References


An influence of notch type on material behaviour under monotonic tension

Tadeusz Szymczak¹, Zbigniew L. Kowalewski², Adam Brodecki³
¹ Centre for Material Testing, Motor Transport Institute
Jagiellonska 80, 03-301 Warsaw, Poland
e-mail: tadeusz.szymczak@its.waw.pl ¹, adam.brodecki@its.waw.pl ³
² Department for Strength of Materials, Institute of Fundamental Technological Research
Pawinskiego 5B, 02-106 Warsaw, Poland
e-mail: zkowalew@ippt.pan.pl

Abstract

The paper presents results of numerical and experimental investigations conducted in order to determine an influence of notch type on material behaviour under monotonic tension. Two kinds of specimens having U and V notches were applied. A digital image correlation system was used to detect variations of strain/stress state components from the beginning of the test up to fracture of the material considered. Assessment of the results led to identification of stress concentration in tips of notches. A comparison of tensile characteristics of unnotched and notched specimens exhibited a great reduction of the yield point independently of geometrical discontinuities applied. In the case of V-notched specimen 50% lowering of elongation was observed.

Keywords: stress concentration, stress distribution, notches, tensile curve, digital image correlation

1. Introduction

Geometrical discontinuities in a form of notches modify magnitudes of stress state components. It is related with their geometrical sizes such as radius, angle or depth. An influence of notches on a material behaviour is usually examined either by theoretical or experimental analysis. The theory enables to assess possible fracture on the basis of the stress concentration factor and to illustrate variations of this parameter as a notch geometry function. In the case of experiment, an influence of geometrical discontinuities on fracture may be determined by the application of various specimen types. Tubular or flat specimens may be distinguished with notches to be cut in the way reflecting special cases of interest. The notches may have various sizes, usually classified as: small [3,5], medium [3] or large [3]. It was found experimentally, that an increase of the notch radius from 0 to 6.35 mm reduces by 50% the number of cycles necessary for an initiation of the crack [1]. The same effect was noticed for an increasing stress concentration factor taking values within a range of 1-2.833 [2,3]. A presence of notches decreases in 60% the stress level enforcing the crack initiation during fatigue tests [5]. The results mentioned above do not sufficiently indicate geometrical discontinuities effect in a zone close to the notch. This problem may be solved using such modern techniques as Digital Image Correlation (DIC). Therefore, the main aim of the paper the application of DIC system for investigation of notches influence on material behaviour during tensile tests carrying out up to the fracture.

2. Experimental procedure

An influence of notch radius on the crack initiation and its subsequent propagation was analysed on the basis of results obtained using Finite Element Method (FEM) and tests performed by means of DIC technique. Flat specimens of 3 mm thickness with three U and V notches having radius equal to 0.25, 1.5 and 2.5 mm and angle 30, 60, 90° respectively were tested, Fig. 1a. A depth of the notches was equal to 1.3 mm. In FEM calculations the specimen geometry was reflected by network of 3D Solid Hex Elements of 0.3 mm high. Calculations were performed for the ideally elastic material. Before the main tests a geometry of discontinuities was checked by the profilometric measurements. Monotonic tensile tests were carried out at room temperature on the servo-hydraulic testing machine at constant displacement velocity equal to 0.5 mm/s. Distribution of the strain components was determined by means of the 4M Aramis Digital Image Correlation system.

3. Results

3.1. Numerical analysis

In order to determine a role of radius and angles of notches, the HMH effective stress distribution in 3D coordinate system was considered, Fig. 1b, c. The results presented on the 0XZ plane did not show any significant differences in the stress distribution due to application of the notches, Fig. 1b.

An opposite result was achieved for the HMH effective stress distribution presented on the 0YZ plane, Fig. 1c. In this case a notch effect is expressed by differences between zones of the maximum stress. The largest area of the stress, expressing the crack occurrence, was obtained for the smallest radius (U specimen) and biggest angle (V specimen). Moreover, a radius increase did not cause variations of the stress level, however, it led to reduction of the maximum stress zone area along the y axis and to its expansion along the z axis. In the case of V specimen the stress reduction, was observed with lowering of a notch angle, Fig. 1c. The effect of the radius and angle of notches on the stress level was also evaluated on the basis of calculation using the equation recommended for notched specimens [4].

As presented in Fig. 2 the stress concentration factor and maximum stress increase linearly with the notch angle increase. In the case of the U notched specimen the largest values of these parameters were obtained for the smallest radius considered. These effects were consistent with the results obtained by means of FEM analysis. The representative results for this type analysis are illustrated in Fig. 1.

149
3.2. Experimental results

The strain distribution obtained under monotonic tension on the multi-notched specimens is shown in Fig. 3a, b.

An effect of the notches is reflected by variations of the HMH effective strain isolines. At the beginning of tension represented by 500th stage, two biggest notches are appeared as significant stress/strain concentrators. Further tension led to the stress/strain components increase in the middle notch, and subsequently, in the largest notch. Finally the specimen fracture appeared (Fig. 3b, c). Comparison of the tensile curves determined for specimens without and with notches enables identification of their essential differences. It is expressed by a clear drop of the yield point observed for test performed on the notched specimen, Fig. 4. Additionally, the material hardening and its instability are clearly manifested. As it is shown in Fig. 4, both curves differ significantly for deformation range higher than 5%.

4. Remarks

The reduction of radius for U-notched specimen leads to the linear increase of the stress concentration factor and maximum stress. The same effect can be observed for the V-notched specimen when the angle takes higher values. The main difference in the shape of the tensile curves for the unnotched and notched specimens takes place in the range of stresses higher than the yield point.

References

Transformation-Induced Creep and Relaxation of TiNi Shape Memory Alloy

Kohei Takeda\textsuperscript{1}, Ryosuke Matsui\textsuperscript{2}, Hisaaki Tobushi\textsuperscript{3}, Elżbieta Pieczyska\textsuperscript{4}

\textsuperscript{1,2,3} Department of Mechanical Engineering, Aichi Institute of Technology
1247 Yachigusa, Yakusa-cho, Toyota, 470-0392, Japan
e-mail: k-takeda@aitech.ac.jp

\textsuperscript{4} Institute of Fundamental Technological Research, Polish Academy of Sciences
Pawinskiego 5B, 02-106, Warsaw, Poland
e-mail: epiecz@ippt.pan.pl

Abstract

If the shape memory alloy (SMA) is subjected to the subloop loading under the stress- or strain-controlled condition, transformation-induced creep or relaxation can appear based on the martensitic transformation (MT). In the design of SMA elements, these deformation properties are important since the deflection or force of SMA elements can change under constant load or constant strain. The conditions for the progress of the MT are discussed based on the kinetics of the MT for the SMA. The creep and relaxation properties are investigated experimentally for TiNi SMA. If the stress is kept constant at the upper stress plateau after loading with a high strain or stress rate up to the stress-holding start strain, the transformation-induced creep occurs due to the spread of the stress-induced martensitic transformation process. If the strain is kept constant at the upper stress plateau after loading with a high stress rate, the transformation-induced relaxation appears due to the exothermic MT until the holding strain and thereafter temperature decreases while holding the strain constant.

Keywords: shape memory alloy, superelasticity, transformation-induced creep, transformation-induced relaxation

1. Introduction

Since the shape memory alloy (SMA) exhibits superior features of an intelligent material, the application of the SMA has drawn the worldwide attention. In the case of the subloop in which strain, temperature and stress vary in the range prior to the martensitic transformation (MT) completion, the starting and finishing conditions of the MT prescribed in the full loop are not satisfied. If the condition of the MT to progress is satisfied, transformation-induced creep and relaxation occurs under constant stress and strain on the upper stress plateau.

In the paper the transformation-induced creep and relaxation in the stress-controlled superelastic subloop loading under a constant strain and constant stress are discussed by the tension test for the TiNi SMA.

2. Materials and specimens

The material used in the tests was a TiNi alloy containing Ti-50.95 at% Ni. This material was produced by Furukawa Techno Material Co., Ltd. and shows SE at room temperature. The thickness and the width of the tape were uniform, equal 0.38 mm and 9.81 mm, respectively. The specimen used in the test was of gage length, 100 mm, where “gage length” (GL) means the distance between the two securing grips.

3. Transformation-induced creep deformation

Figure 1 shows the stress-strain curve obtained from the creep test under a constant stress rate of 5 MPa/s up to a strain of 2\% at the upper stress plateau, followed by a constant stress. In Fig. 1, the stress-induced martensitic transformation (SIMT) starts at a strain of 1.3 \% (point S\text{\textsubscript{0}}) in the loading process under a constant stress rate. If stress is controlled so as to remain constant at its level for 2\% strain (point H\textsubscript{1}), it initially

Figure 1: Stress-strain curve under a stress rate of 5 MPa/s followed by holding constant stress at 508 MPa from a strain of 2\% (point H\textsubscript{1}) during loading

Figure 2: Thermograms of temperature distribution on the specimen surface under a stress rate of 5 MPa/s followed by holding constant stress from a strain of $\varepsilon_1 = 2\%$ during loading
fluctuates slightly before settling down to a constant 508 MPa at a strain of 3.5% (point $C_1$). Next the strain is increased to about 8% (point $F_1$). This phenomenon of strain increase under a constant stress is similar to the normal creep deformation. The explanation in this case would be that the SIMT causes the temperature to increase during loading up to a strain of 2%, after which it decreases under a constant stress. Conditions are therefore satisfied for the SIMT to progress and strain increases.

Figure 2 shows thermograms of temperature distribution on the surface of a specimen. As can be seen from Fig. 2, the SIMT process due to the exothermic reaction first appears at two ends during loading at a strain level of 2%, and then spreads towards the center where the bands combine into one, completing the SIMT. When the stress is held constant at the level reached for a 2% strain, the SIMT bands spread due to a temperature decrease. Transformation heat is generated at each new point of advance in the SIMT process, which leads to a chain reaction in the SIMT, resulting in transformation-induced creep deformation.

4. Transformation-induced relaxation

The stress-strain curve obtained by the relaxation test under a stress rate of 5 MPa/s until a point $H_1$ at a strain $\varepsilon_1 = 6\%$ followed by holding the strain $\varepsilon_1$ constant is shown in Fig. 3. As can be seen in Fig. 3, in the strain holding process at $\varepsilon_1 = 6\%$, the stress decreases from $\sigma_1$ to $\sigma_2$, resulting in transformation-induced relaxation $\Delta \sigma = \sigma_2 - \sigma_1$.

Figure 4 shows temperature distribution on a specimen surface at various strains during loading and at various stresses at a constant strain obtained by the thermography. Figure 5 shows the relationship of stress $\sigma$ and temperature change $\Delta T$ between the average temperature on the specimen surface and the atmosphere temperature with time $t$ during loading and holding the strain constant. As can be seen in Figs. 4 and 5, in the loading process from the MT start point $S_0$ to the point $H_1$, the strain rate becomes high and little time is left for the heat generated due to the exothermic MT to transfer to the atmosphere air, resulting in a temperature increase of the specimen. In the strain holding stage from points $H_1$ to $H_2$, temperature decreases by the air and the condition for the transformation to progress is satisfied, resulting in the progress of the MT. As a result, stress relaxation appears during a constant strain test.

Figure 3: Stress-strain curve under a stress rate of 5 MPa/s till a point $H_1$ at a strain $\varepsilon_1 = 6\%$ followed by holding the strain $\varepsilon_1$ constant

![Stress-strain curve](image1)

Figure 4: Thermograms of temperature distribution on the specimen surface under a stress rate of 5 MPa/s till a point $H_1$ at a strain $\varepsilon_1 = 6\%$ followed by holding the strain $\varepsilon_1$ constant

![Thermograms](image2)

Figure 5: Variation in stress $\sigma$ and average temperature change $\Delta T$ on specimen surface in the relaxation test

As can be seen in Fig. 5, temperature varies significantly in the early stage during a constant strain phase and thereafter saturates a certain value. Corresponding to this temperature change, stress relaxation appears markedly in the early stage during the constant strain test.

5. Conclusions

The transformation-induced creep and relaxation under the stress-controlled subloop loading in TiNi SMA tape were investigated based on evidence of local temperature variation as measured by infrared thermography during the creep and relaxation tests. The results obtained can be summarized as follows.

1. If the stress is kept constant at the upper stress plateau after loading up to the stress-holding start strain under a constant stress rate, the transformation-induced creep deformation occurs due to the spread of the SIMT process.

2. If the strain is kept constant at the upper stress plateau after loading with a stress rate, the transformation-induced relaxation appears due to the exothermic MT until the holding strain and thereafter temperature decreases while holding the strain constant.
Influence of Nitrogen Ion Implantation on Fatigue of a TiNi Shape Memory Alloy Tape

Kohei Takeda¹, Ryosuke Matsui², Hisaaki Tobushi³, Neonila Levintant-Zayonts⁴, Stanislaw Kucharski⁵

¹,²,³ Department of Mechanical Engineering, Aichi Institute of Technology
1247 Yachigusa, Yakasa-cho, Toyota, 470-0392, Japan
e-mail: k-takeda@aitech.ac.jp

⁴,⁵ Institute of Fundamental Technological Research, Polish Academy of Sciences
Pawińskiego 5B, 02-106, Warsaw, Poland
e-mail: skuchar@ippt.pan.pl

Abstract

The fatigue property of TiNi shape memory alloy (SMA) is one of the most important subjects in view of evaluating functional characteristics of SMA elements. In the paper, the nitrogen ion implantation (NII) was applied into a TiNi SMA tape, in which both flat surfaces were ion-implanted from two opposite directions and the influence of NII on the bending fatigue life was investigated. The larger the bending strain amplitude, the shorter the fatigue life is. The fatigue life of the NII-treated tape is longer than that of the non-implanted tape. The fatigue crack nucleates at the central part of the flat surface of the non-implanted tape. The fatigue crack nucleates at the corner on the surface near the flat surface of the NII-treated tape.

Keywords: shape memory alloy, titanium-nickel alloy, superelasticity, nitrogen ion implantation, fatigue, bending

1. Introduction

The shape memory alloy (SMA) is expected to be applied as intelligent material since it shows the unique characteristics of the shape memory effect and superelasticity (SE). In the growing number of TiNi SMA applications, these materials should fulfill high requirements of fatigue, corrosion and wear resistance. On the other hand, the application of SMA has some limitations, particularly in thermomechanical cyclic loading cases, when structural components can be damaged due to fatigue. In this case, fatigue of SMA is one of the important properties in view of evaluating functional characteristics as SMA elements.

In the previous paper, the nitrogen ion was implanted into TiNi SMA wires from two opposite directions and the influence of ion-implantation treatment on the bending fatigue properties was investigated. It was confirmed that the fatigue life becomes longer by the ion implantation. However, in the case of wires, ion is not uniformly implanted in the cylindrical surface layer of the wire.

In the paper, the influence of nitrogen ion implantation (NII) on the tensile deformation, bending fatigue life and fatigue fracture surface was investigated for TiNi SMA tapes.

2. Transformation temperature

The TiNi SMA tape of a width of 2.5 mm and a thickness of 1.0 mm was ion-implanted on both flat surfaces from two opposite directions by nitrogen ion beam with acceleration energy of 50 keV. The total doses of implanted ion were 8 × 10¹⁶, 3 × 10¹⁷ and 2.5 × 10¹⁸ ions/cm². The DSC thermograms for non-implanted and ion-implanted with 2.5 × 10¹⁸ ions/cm² tapes are shown in Fig. 1. If the nitrogen ion was implanted, the transformation temperatures increase a little. If the shape-memory processing temperature is high, the phase transformation temperature increases due to the temperature rise during the ion implantation process.

![Figure 1: DSC thermograms for two kinds of tapes with non-implanted and ion-implanted with 2.5 × 10¹⁸ ions/cm²](image-url)
fatigue test. In Fig. 4, the nitrogen ion is implanted, the transformation temperatures decrease and the partial SE appears in place of the SE.

With 2.5 × 10^{18} ions/cm^2 tapes obtained by the tension test at room temperature, the bending fatigue life becomes longer. As the bending strain amplitude increases if NII is treated not only on the surface at the maximum stress point but also on the surface in the region near the maximum stress point, the larger the bending strain amplitude, the shorter the fatigue life is. The fatigue life of the NII-treated tape is longer than that of non-implanted tape.

The fatigue crack nucleates at a certain point F_c in the central position of the flat surface and propagates towards the center in an ellipsoidal pattern. In the case of the ion-implanted with 2.5 × 10^{18} ions/cm^2 tape, the crack nucleates at a corner of the tape and propagates towards the center and along the flat surface of the tape with the higher speed of progression.

Figure 4 shows SEM photographs of a fracture surface of two kinds of tapes obtained by the alternating-plane bending fatigue test; (a) Non-implanted, (b) Implanted with 2.5 × 10^{18} ions/cm^2.

6. Conclusions

The nitrogen ion was implanted into TiNi SMA tapes, in which both flat surfaces were ion-implanted from two opposite directions, and the influence of NII on the fatigue life of alternating-plane bending was investigated. The results obtained are summarized as follows:

1. The larger the bending strain amplitude, the shorter the fatigue life is. The fatigue life of the NII-treated tape is longer than that of non-implanted tape.
2. The fatigue crack nucleates at the central part of the flat surface of non-implanted tape. The fatigue crack nucleates at the corner on the surface near the flat surface of the NII-treated tapes.
3. In practical applications of SMAs, the fatigue life of SMA elements increases if NII is treated not only on the surface at the maximum stress point but also on the surface in the region near the maximum stress point.

5. Fatigue fracture surface

Figure 4 shows SEM photographs of a fracture surface of two kinds of tapes obtained by the alternating-plane bending fatigue test. In Fig. 4, F_c denotes the initiation point of the fatigue crack. In the case of the non-implanted tape, the crack nucleates.
Numerical and experimental analysis of a cool thermal storage unit

Mateusz M. Zając¹, Jarosław Karwacki², Roman Kwidziński³
¹,²,³ The Szewalski Institute Of Fluid-Flow Machinery, Polish Academy of Sciences, Fiszera 14, 80-231 Gdańsk, Poland
e-mail: mzajac@imp.gda.pl, jkarwacki@imp.gda.pl, rk@imp.gda.pl

Abstract

The subject of this study is a latent heat thermal energy storage unit which was made from a plate heat exchanger filled with a Phase Change Material. The unit was designed to operate with a refrigeration installation to stabilize the cooling power output during compressor off-time. Numerical calculations were performed using ANSYS Fluent software. Results of the calculations exhibit high dependency on assumptions regarding material properties. Therefore experimental study was performed for validation purposes.

Keywords: phase change material, thermal energy storage, numerical simulation, experiment.

1. Introduction

Latent Heat Thermal Energy Storage (LHTES) units can effectively rise thermal inertia of cooling or heating installations thus minimizing the fluctuations of the controlled temperature [3]. Using Phase Change Material (PCM) with high latent heat capacity allows for storage of large amounts of thermal energy [4]. It rises the problem of an effective heat exchange with the Heat Transfer Fluid (HTF) [2]. In order to compensate for low thermal conductivity of the PCMs the plate heat exchanger with high area to volume ratio has been chosen for a base of a latent heat storage.

2. Simulations

The computational geometry, shown in Fig. 1, represents a segment of a thermal energy storage unit constructed on a base of a plate heat exchanger. The HTF flows between the layers of PCM exchanging heat through a steel plate.

Figure 1: Computational geometry

The simulations were conducted in ANSYS Fluent 15.07 [1] using several meshes with number of elements ranging from $1 \times 10^5$ to $5 \times 10^5$. The density of the mesh did not influence the results of the simulation. Due to periodicity of the domain, symmetry boundary conditions were set on the sides of the mesh. The plate of the heat exchanger was modeled as a zero-thickness internal wall with coupled thermal boundary condition applied simulating the 0.4 mm thick steel plate.

At the inlet of the HTF velocity $v_{in}$ was set as a boundary condition while outlet is modeled as an outflow. External walls of the model were adiabatic.

2.1. Material properties

Properties of materials used in the latent heat storage unit provided by the manufacturers are presented in Tab. 1. However, properties of PCM were given only for the solid phase. Although the temperature range used in the simulations was narrow, high change of thermal parameters of the PCM occurs as it passes from one phase to the other. In order to take this phenomena into account, the thermal conductivity and specific heat of the PCM in the simulations were defined as temperature-dependent piecewise linear functions. Their highest parameter gradient was set in the phase change temperature range $\Delta T_p$. For reference, simulations with constant material properties were conducted as well.

2.2. Calculations settings

At the starting point of calculations temperature throughout the whole domain was set constant. HTF flow was calculated as a fully developed and constant in time. Time step of 0.02 s was used for transient calculations of heat exchange and phase change, which were the second step. Almost twenty cases, with various values of HTF inlet temperature $T_{in}$, $\Delta T_p$, $v_{in}$ and phase material properties were simulated.
3. Results

It was found that $\Delta T_p$ has the biggest influence on the HTF temperatures at the outlet of the latent heat storage unit. Thus, in order to provide accurate simulations, this parameter has to be precisely defined. When $\Delta T_p = 0$, the numerical model did not take into account the latent heat of phase transition resulting in rapid equalization of temperature in the calculation domain shown in Fig. 2. Due to uncertainty of the correct $\Delta T_p$ value, the numerical results were validated experimentally shown in Fig. 3 and Fig. 4. Higher HTF flow rates resulted in shortening of the PCM cooling time. In cases where $T_w$ was given in a form of a time dependent dropping linear function, the process similar to Differential Scanning Calorimetry was observed and the outlet HTF temperature profile clearly indicated the phase-change starting point shown in Fig. 5.

Figure 2: Response of HTF outlet temperature to step fall of inlet temperature for various values of inlet velocity $v_{in}$ [m/s] and phase change temperature range $\Delta T_p$ [K]

Figure 3: Photography of the experimental LHTES unit

Figure 4: Scheme of the experimental LHTES installation

Figure 5: Profile of outlet HTF temperature when the inlet temperature decreases linearly with time

References


Frequency-dependent temperature and strain evolutions of NiTi wire during cyclic stress-controlled martensitic transformation

Lin Zheng¹, Yongjun He², Ziad Moumni³

¹,²,³ Unité de Mécanique - Groupe de Recherche Matériaux et Structures, ENSTA ParisTech
828, boulevard des Maréchaux 91120 Palaiseau, France
e-mail: lin.zheng@ensta-paristech.fr ¹, yongjun.he@ensta-paristech.fr ², ziad.moumni@ensta-paristech.fr ³

3 Northwestern Polytechnical University
Xi’an 710072, P.R.China
e-mail: ziad.moumni@ensta-paristech.fr

Abstract

Both temperature and strain evolutions were examined during the cyclic stress-controlled phase transformation on a trained superelastic NiTi wire at a wide frequency range (f = 0.0002 – 1 Hz). A strong frequency-dependent coupling between the temperature variation and the stress-strain evolution was observed: at low and intermediate frequencies, the wire mean temperature shifts downwards with the cycle number n and the maximum temperature increases; at high frequencies, the maximum strain decreases during cycling, while the mean temperature shifts upwards. For the steady-state cycles, the maximum strain decreases monotonically with increasing f, indicating that the martensitic transformation is not complete at intermediate and high frequencies; accordingly the stress-strain hysteresis-loop area (damping capacity) varies non-monotonically with f.

Keywords: NiTi shape memory alloy, frequency dependence, stress-controlled tension, cyclic martensitic transformation, temperature and strain coupling

1. Introduction

NiTi shape memory alloys (SMAs) are employed in various applications where the SMA structures are required to sustain cyclic loading to a certain number, from tens to several millions [1,3], and usually at a wide range of loading frequency [1,5]. So the material’s frequency-dependent cyclic behaviors are of great importance. Despite of the increasing research attentions on the thermo-mechanical coupling [4,5], the firsthand experimental evidences on the coupled thermal and mechanical evolutions under stress-controlled cyclic phase transformation are still lacking.

2. Experiments and discussion

The material used in this study was a commercial superelastic NiTi polycrystalline wire (Johnson Matthey Inc. USA) of a diameter d = 1.57 mm with a composition 55.92 wt.% Ni; the austenite finish temperature A_f is about 10 ± 5°C. The overall length of the wire specimen was 60 mm, and the gauge length L_0 between two grips was about 35 mm. The stress-controlled cyclic tensile tests of σ_max = 400 MPa at 12 frequencies ranging from 0.0002 Hz to 1 Hz were conducted on a dynamic testing machine (Instron ElectroPuls E3000). An ultra-thin K-type thermocouple with a diameter of 0.025 mm was attached to the wire to measure its surface temperature. Before the tests, the fresh specimen was trained under strain-controlled cyclic deformation (ε_max = 5.6%) for 100 cycles at frequency f = 0.01 Hz.

2.1. Temperature variation and stress-strain curve evolution

Figure 1(a) shows the results at low frequency f = 0.0002 Hz where the maximum and minimum temperature variations of the specimen were respectively ΔT_max ≡ T_max – T_ambient = 5.3°C and ΔT_min ≡ T_min – T_ambient = –4.3°C during the forward/reverse phase transformation. At this low-frequency cyclic loading, in a sufficient time for the heat exchange between the specimen and the ambient in a cycle, the temperature variation and the stress-strain curve didn’t change with the cycle number n. It can be seen that the stress-strain curve is close to the isothermal one (obtained from a strain-controlled tensile test at f = 0.0005 Hz), but it has a smooth hardening rather than a rough stress plateau during the phase transformation.

The results of a typical intermediate frequency (f = 0.05Hz) are shown in Fig. 1(b) where both the temperature variation and the stress-strain curve changed with n until n > 8. In the transient stage (n = 1 – 8), both the temperature variation and the stress-strain curve shifted downwards; particularly, the amplitude of the temperature variation ΔT_ambi ≡ ΔT_max – ΔT_min increased from 18.1°C in the first cycle to 19.1°C in the steady-state cycles, in accordance with the increase of the maximum strain ε_max by 0.30% — larger transformation strain (volume) releases more latent heat thus causing larger temperature variation. In a cycle at this intermediate frequency, the heat exchange between the specimen and the ambient was not fast enough to take out the total latent heat generated during the forward phase transformation, so the specimen temperature increased significantly, leading to a large hardening in the stress-strain curve. At the loading end, the released latent heat of the partial phase transformation was able to increase the temperature high enough to make the transformation stress equal to the controlled maximum stress (σ_max = 400 MPa). Therefore, the value of ε_max was much smaller than that at f = 0.0002 Hz in Fig. 1(a).

When f was increased to 1 Hz (see Fig. 1(c)), more cycles were required to reach the steady state. The specimen temperature was always higher than the ambient temperature,
and the mean temperature variation $\Delta T_{\text{mean}} = \frac{\Delta T_{\text{max}} + \Delta T_{\text{min}}}{2}$ increased with the cycle number $n$. Opposite to the trends at $f = 0.05$ Hz in Fig. 1(b), at this high frequency both $\Delta T_{\text{max}}$ and $\Delta T_{\text{min}}$ increased with $n$, and $\Delta T_{\text{ampl}}$ decreased from 10.4°C in the first cycle to the 7.4°C in the steady-state cycles, conforming to the decrease of $\varepsilon_{\text{max}}$ by 0.31%.

2.2. Dependence of the steady-state responses on frequency

Figure 2: Dependence of the steady-state output strain $\varepsilon_{\text{max}}$ and the steady-state hysteresis-loop area $D'$ on frequency $f$

The steady-state output strain $\varepsilon_{\text{max}}$ and the hysteresis-loop area $D'$ for all the 12 frequencies are summarized in Fig. 2. It is seen that $\varepsilon_{\text{max}}$ decreases monotonically with increasing $f$, indicating that the complete martensitic transformation occurs at low frequencies (e.g., $f = 0.0002$ Hz in Fig. 1(a)) while only partial transformation takes place at intermediate and high frequencies. And the steady-state hysteresis-loop area $D'$ varies non-monotonically with $f$, reaching the peak at an intermediate frequency ($f = 0.002$ Hz). Such frequency dependence is important for damping applications and fatigue research.

3. Summary

Both the temperature variation and the stress-strain evolution during the cyclic stress-controlled martensitic transformation strongly depend on the loading frequency. This frequency dependence is mainly due to the coupling among the temperature-dependent transformation stress, the effects of the latent heat, the mechanical dissipation and the heat exchange with the environment. Our experiments show the following frequency dependences:

- In transient cycles, at low and intermediate frequencies, the specimen mean temperature shifts downwards with the cycle number $n$ and the maximum strain increases; at high frequencies, the maximum strain decreases during cycling, while the mean temperature shifts upwards.

- In steady-state cycles, the output strain decreases monotonically with increasing frequency; the hysteresis-loop area varies in a non-monotonic way, reaching the peak at an intermediate frequency, which is similar to the trend under strain control [2,4].

References


MS06

Isogeometric Analysis and Applications

organized by Z. Kacprzyk
Available numerical implementations of isogeometric analysis

Zhigniew Kacprzyk¹, Katarzyna Ostapska-Łuczewska²
¹Faculty of Civil Engineering, Warsaw University of Technology
al. Armii Ludowej 16, 00-637 Warsaw, Poland
e-mail: z.kacprzyk@il.pw.edu.pl
²Faculty of Civil Engineering, Warsaw University of Technology
al. Armii Ludowej 16, 00-637 Warsaw, Poland
e-mail: k.ostapska@il.pw.edu.pl

Abstract

The subject of the article concerns Isogeometric Analysis (IGA) as a new formulation within Finite Element Method [1]. A motivation for this new approach was presented together with theoretical foundations of the method. Special focus was put on the new concept of element and new geometry data interpretation in terms of analysis.

In this paper the problems of implementation of the IGA method is discussed. Existing FEM codes do not allow for easy adaptation of the IGA. Currently there are three packages dedicated to the IGA: GeoPDEs, Abaqus plugin and Rhino-Grashopper-Matlab, [3, 2, 4]. Geometry and mesh transfer from Rhino software with Grasshopper plug-in was also described in this paper. Finally, authors conclusions concerning new approach to finite element method were presented.

Keywords: Isogeometric Analysis, NURBS, B-splines, Finite Element Method, Computer Aided Design, Computer Aided Engineering

1. Introduction

Most of recent approaches in the field of finite element method focus on solving certain difficulties for a narrow class of problems. Isogeometric Analysis has an entirely different motivation and takes into account the FEM and analyse problems as a whole. The authors of this method [1] took industry needs and problems into consideration and came with the solution that aims to integrate geometry description used in CAD systems with one applied during FEM analysis. It would appear that the topic was already covered by a widely developed market of mesh generators and geometry transfer tools. However, this new approach makes geometry transfer needless and introduces full automation that answers problems with a design process bottleneck in CAD-CAE industry.

The subsequent change in IGA element concept in comparison to FEM element concept demands thorough and detailed study of CAD geometry representation both from mathematical and numerical point of view. It is a necessary knowledge for a designer for an intuitive engineering approach.

2. Method

2.1. CAD systems and geometry description

Computer Aided Design systems are now widespread design tool in every branch of industry. CAD model can be created in many different ways, with use of different file formats, geometry description, algorithms etc. Some of CAD designs are transferred at different stages of design process to the Computer Aided Engineering systems. The latter use Finite Element Analysis most commonly to perform for example static structural, thermal, dynamic or hydrodynamic analysis on the provided geometry. From academic point of view the formal transfer from CAD to CAE may seem irrelevant issue but it has become a huge problem recently. The main obstacle is the proper model for FEA preparation out of data received from CAD stage of design process. It is estimated in many companies and research institutes that 80% of total FEA time for the product is related to the input geometry adjustment. This encompasses simplifications in the CAD model, partitioning, mesh generation and necessity to repeat those operations in case of a resulting model unsuitable for analysis.

2.2. B-Splines and NURBS

We create Non Uniform Rational B-Spline’s (NURBS) applying certain basis function. One of the curve parameter is a knot vector, [1].

\[ \Xi = \{\xi_1, \xi_2, \ldots, \xi_{n+p+1}\}, \]

where: \(\xi_i \in \mathbb{R}\) is the \(i^{th}\) knot, \(n\) - number of basis functions, \(p\) is the polynomial order.

NURBS generalize the B-Spline’s, defined as (for \(j = 0\)),

\[ N_{i,0}(\xi) = \begin{cases} 1 & \text{if } \xi_i \leq \xi \leq \xi_{i+1}, \\ 0 & \text{otherwise}. \end{cases} \]  

For \(j = 1, 2, 3, \ldots\), they are defined by

\[ N_{i,j}(\xi) = \frac{\xi - \xi_i}{\xi_{i+j} - \xi_i} N_{i,j-1}(\xi) + \frac{\xi_{i+j+1} - \xi}{\xi_{i+j+1} - \xi_{i+1}} N_{i+1,j-1}(\xi) \]

We present NURBS curves basis functions as following [5],

\[ R^p_i(\xi) = \frac{N_{i,p}(\xi)w_i}{\sum_{j=1} N_{j,p}(\xi)w_j} \]

where: \(R^p_i(\xi)\) - NURBS curve basis function, \(N_{i,p}(\xi)\) - B-Spline basis function, \(w_i\) - weight.
The NURBS curves are constructed by a linear combination of basis functions. Basis functions are defined by coefficients, which are control points coordinates. Polyline joining control point is called control polygon. Formula defining NURBS curves in space is in the form:

\[ C(\xi) = \sum_{i=1}^{n} R_p^i(\xi) B_i \]  

where \( C(\xi) \) - coordinates of curve in space, \( R_p^i(\xi) \) - basis function, \( B_i \) - coordinates of the curve control points.

3. IGA algorithm description and software

In Isogeometric Analysis an element is defined by knot spans of the parametric domain. Element nodes are control points not being interpolation points (except in special cases). A number of degrees of freedom for one element is the number of control points times number of degrees of freedom for each control point. Figure 1 shows elements of the marked nodes (upper figures) and complex structure (bottom figure). It is therefore a different way of combining elements than in the classical method. This difference means that the existing computing finite elements systems cannot be used. Some problem is to import geometry from a CAD system and its division into finite elements. Building from the beginning of the computing system is time and cost-consuming.

The IGA algorithm numerical implementation is very strongly supported in the B-Spline/NURBS evaluating algorithms. The algorithm is more complicated than the classical FEM algorithm at the initial stage due to the full CAD geometry definition incorporation.

Therefore, the authors are trying to use some software tools for each step of the algorithm. Examples of such a procedure are GeoPDEs [3]. A different approach is proposed by the authors of the tool-kit to the system Abaqus [2].

4. Examples

The present example concerns temperature field in the complex geometry (Fig. 2) with assumed boundary conditions on selected edges, [4]. In this study, we used the packages: Rhino, Grasshopper and Matlab. The Results show that while in simple problem of steady state thermal analysis minimal amount of elements is required to obtain good results with respect to the quantity, a refinement is necessary for geometry representation. The results from Matlab were compared with those obtained from Ansys FEM system.

5. Summary

Isogeometric Analysis proves effective in case where exact geometry representation plays a crucial role. The new approach to the FEM gives us a possibility to describe geometry, the construction of a global matrix and defining boundary conditions in different way. The new method, however, requires creating program codes from scratch. The authors analysed the new available software and tried to identify the best solution.

References

Comparison of IGA and FEM for the Poisson benchmark PDE

Marcin Łuczkowski¹, Katarzyna Ostapska-Łuczkowska², Witold Cecot³

¹,²Faculty of Civil Engineering, Cracow University of Technology
Warszawska 24, 31-155 Cracow, Poland
e-mail: luczkowski.marcin@gmail.com

²Department of Structural Engineering, NTNU Norges teknisk-naturvitenskapelige universitet
Høgskoleringen 1, 7491 Trondheim, Norway

Abstract

The paper concerns the analysis of the Poisson problem with an Isogeometric approach to Finite Element Method. The theoretical basis of geometry description using NURBS was presented together with practical examples. The article is focused on the problem of creating and transferring h-refinement from CAD software to IGA solver, implemented by the authors in Matlab environment. The geometry has been created in Rhinoceros and the transferring algorithm was written in Grasshopper plug-in. The code written in Matlab directly uses the CAD-created geometry. Convergence of IGA approximation of the Poisson problem solution of IGA was performed with the FEM results.

Keywords: Isogeometric Analysis (IGA), Finite Element Method (FEM), Poisson Equation, NURBS, Computer Aided Design (CAD)

1. Introduction

The Isogeometric approach to Finite Element Method (IGA) is currently under intensive development. It was proposed as an answer to the problem of dealing with more complicated CAD-based geometry in the FEA (Finite Element Analysis) industry. Nowadays, most of graphical programs provide a wide range of geometric description. A well-known B-Spline is now used as commonly as the non-uniform B-Spline polynomial (shortly NURBS). The IGA gives a possibility to engineers and scientists to use sophisticated, curvilinear geometry in the FEM analysis without losing any information about it.

2. NURBS geometry

The Non Uniform Rational B-Spline is the generalization of the B-spline curve, that is defined by the Cox-de Boor recuration formula:

for \( p = 0 \):

\[
N_{i,0}(\xi) = \begin{cases} 1 & \text{if } \xi_i \leq \xi < \xi_{i+1} \\ 0 & \text{otherwise} \end{cases}
\]  

(1)

for \( p = 1, 2, 3 \ldots \):

\[
N_{i,p}(\xi) = \frac{\xi - \xi_i}{\xi_{i+p} - \xi_i} N_{i, p-1}(\xi) + \frac{\xi_{i+p+1} - \xi}{\xi_{i+p+1} - \xi_{i+1}} N_{i+1, p-1}(\xi)
\]  

(2)

Wider possibilities of geometry description were obtained adding the weights the control points. Values of weights \( w_i \) are changing the power of attraction of the control point to the curve in its vicinity. The NURBS basis functions are given by:

\[
R^p_i(\xi) = \frac{N_{i,p}(\xi)}{\sum_{j=1}^{n} N_{j,p}(\xi)w_j}
\]  

(3)

Using equation (3) with the control points coordinates \( B_i \) leads to the equation for a NURBS curve:

\[ C(\xi) = \sum_{j=1}^{n} R^p_i(\xi)B_i \]  

(4)

Today almost every CAD software includes a NURBS drawing tool. In the work the Rhino 5 was applied to create geometry. To define a NURBS curve there is a need to declare the polynomial degree \( p \) and choose the control points. In order to control the weight value, the algorithm in Grasshopper plug-in was created.

Figure 1: Bspline and NURBS surfaces comparison

In order to appreciate the possibility which gives us NURBS technology see Fig. 1. Adding weight \( 1/\sqrt{2} \) to the central control point a perfect circular curve is obtained, impossible using Bsplines only. Until the NURBS were introduced, creating circular curves was possible using the cylindrical coordinates or a circle equation only.

3. h-refinement

Most of the present day FEM software gives a possibility of creating mesh refinement inside the program. Because of the
Table 1: Numerical results of the Poisson equation problem

<table>
<thead>
<tr>
<th></th>
<th>IGA analysis</th>
<th></th>
<th>CALFEM analysis</th>
<th></th>
<th>HP2D analysis</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Nodes</td>
<td>Temperature [°C]</td>
<td>Nodes</td>
<td>Temperature [°C]</td>
<td>Nodes</td>
<td>Temperature [°C]</td>
</tr>
<tr>
<td>1</td>
<td>9</td>
<td>0.2418</td>
<td>16</td>
<td>0.2211</td>
<td>15</td>
<td>0.2522</td>
</tr>
<tr>
<td>2</td>
<td>30</td>
<td>0.2515</td>
<td>64</td>
<td>0.2427</td>
<td>45</td>
<td>0.2501</td>
</tr>
<tr>
<td>3</td>
<td>90</td>
<td>0.2505</td>
<td>400</td>
<td>0.2488</td>
<td>153</td>
<td>0.2500</td>
</tr>
<tr>
<td>4</td>
<td>306</td>
<td>0.2501</td>
<td>1600</td>
<td>0.2497</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>5</td>
<td>1224</td>
<td>0.2500</td>
<td>3600</td>
<td>0.2499</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>6</td>
<td>4290</td>
<td>0.2500</td>
<td>4800</td>
<td>0.2499</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

NURBS formula decision was made to create mesh in a graphic program and export the control points as well as weights directly to the IGA solver. Creating and controlling the mesh in this procedure is simpler and more intuitive than by the automatic algorithms built in most of the FEM software. A refined mesh is shown in Fig. 2.

4. Poisson equation

In order to verify the IGA code and the h-refinement process a well known benchmark is solved. This problem is described by the Poisson second order partial differential equation (5) in 2D.

\[-\nabla^2 u(x, y) = f(x, y) \quad u = 0 \quad \forall \quad (x, y) \in \Omega\] (5)

The \(f(x, y)\) was assumed as:

\[f(x, y) = \frac{8 - 9 \sqrt{x^2 + y^2}}{x^2 + y^2} \sin(2 \arctan(\frac{y}{x}))\] (6)

Analytical solution of the equation for the central point of the geometry shown in Fig. 3. equals \(-0, 25^\circ C\).

5. Convergence

Results of the minimum temperature for the IGA and FEM were shown in Table 1. The FEM analysis was performed with linear (CALFEM) and bilinear (HP2D [3]) shape functions. Convergence of the solution infinum approximation is shown in Fig. 4.

6. Conclusions

The most significant advantage of the IGA approach is the possibility to make numerical simulations on exactly the same geometry as created in CAD software. The IGA convergence was between the rules offered by the 1st and 2nd order FEM. Further tasks will be performed to obtain some general conclusions. Mesh preparation is easy and with h-refinement we can obtain regular elements. The IGA can be viewed as a generalization of FEM to include CAD in FEA framework without approximation in the way easy to implement way. This poses the problem of recoding the existing commercial software to take that into account. Both pre and post-processors should be created with a special treatment of the NURBS characteristics. The disadvantage of this method is also the non-intuitive element description. Further convergence tests in different norms will be presented during the conference.

References

LBM, Meshless and Related Methods in Computational Fluid and Solid Mechanics
– a Session in Honor of Prof. Janusz Orkisz in connection with His 80th Birthday
and in the recognition of important scientific achievements in Mechanics

organized by H. Kudela, J. Pozorski, J. Rokicki, K. Szewc and J. Szumbarski
A mesh-free particle model for simulation of trimming of aluminum alloy sheet

Łukasz Bohdal¹, Radosław Patyk²

¹,²Faculty of Mechanical Engineering, Koszalin University of Technology
Raclawicka 13-17, 75-638 Koszalin, Poland
e-mail: lukasz.bohdal@tu.koszalin.pl¹, radoslaw.patyk@tu.koszalin.pl²

Abstract

In the paper, the applications of mesh-free SPH (Smoothed Particle Hydrodynamics) continuum method to the simulation and analysis of trimming process of aluminum alloy sheet is presented. Dealing with trimming simulations the existing literature solutions apply finite element method (FEM) to analysis of this process. The approach presented in the work and its application to trimming of aluminum alloy sheet allows for a complex analysis of physical phenomena occurring during the process without significant deterioration in the quality of the finite element mesh during large deformation. This allows for accurate representation of the loss of cohesion of the material under the influence of cutting tools. An analysis of state of stress, strain and fracture mechanisms of the material is presented. The influence of process parameters: clearance, blade sharpness and cutting angle on burr formation and quality of cut surface is investigated. In experimental studies, an advanced vision-based technology based on digital image correlation (DIC) for monitoring the cutting process is used.

Keywords: trimming, mesh-free particle model, SPH, aluminum alloy

1. Introduction

Shear plays an important role in mechanical manufacturing area, the nature of trimming being concerned with many related subjects of technology and industry. However trimming is a very complicated process. There are several factors, such as nonlinearities (geometrical, physical and thermal), large deformation, friction, large strain rates of material and material characteristics that have a direct influence on the process and are sensitive to each other. Knowledge of the trimming process is based mainly on experimental methods, which are often expensive and unable to be extrapolated to other cutting configurations. Trimming modelling becomes an increasingly important tool in gaining understanding and improving this process. At the moment the trimming numerical models are based on Lagrangian or Arbitrary Lagrangian Eulerian (ALE) Finite Element Methods (FEM). Hence, the use of grid/mesh can lead to various difficulties in dealing with problems with free surface, deformable boundary, moving interface, and extremely large deformation and crack propagation. The mentioned disadvantages of the finite element models can be eliminated using following mesh-free methods: smoothed particle hydrodynamics (SPH), element-free Galerkin method, discrete element method, amongst others. In this paper, the applications of mesh-free SPH methodology to the simulation and analysis of 3-D trimming process is presented. The developed model is used to analysis of residual stresses in workpiece during and after process under different conditions. The influence of process parameters: clearance, blade sharpness and cutting angle on burr formation and quality of cut surface is investigated. Next, the model is validated with experimental research by using vision-based solutions.

2. SPH method

The SPH is a total Langrangian and is a truly mesh-free technique initially developed by Gingold and Monaghan [1] and others for the analysis and simulation of astrophysics problems. The technique was later extended to model of discontinuous flows with large deformations and to analysis and simulate for large strain solid mechanics problems. The SPH combines the advantages of mesh-free, particle methods and Langrangian. The advantage of the mesh-free method is its capability to solve problems that cannot be influencely solved using other numerical approaches.

The main differences between FEM and SPH are the absence of a grid and in the discretisation of continuum. It does not suffer from the mesh distortion problems that limit Lagrangian methods based on structured mesh when simulating large deformations [2]. The smoothing of field variables is performed in the area with radius \( h \), called the smoothing length, over which the variables are smoothed by a kernel function. This means that the value of a variable in any spatial point can be obtained by adding the relevant values of the variables within two smoothed lengths.

The SPH approximation of the equation for continuum mechanics uses the following approaches. A function \( f(x) \) is substituted by its approximation \( A(x,h) \), characterising a body condition. For example, the velocities of body points in a particular area are approximated with the following expression:

\[
A_j(x,h) = \int f(y) \cdot W(x,h) dy,
\]

where \( W(x,h) \) is a smoothed kernel function.

Figure 1: Comparison between FE and SPH modelling (9 elements) [2]
3. Coupled SPH+FEM model and sample of results

In the suggested approach to the modeling of the trimming process, a coupling of the FEM model and the model based on hydrodynamic particles (the SPH method) has been proposed. A three-dimensional model of trimming is built in LS - Dyna solver. Numerical calculations are performed for the 3D state of strain and 3D state of stress in this model.

AA6111-T4 aluminum alloy is used as the material to be cut in the numerical and experimental studies, with thickness of \( t = 1 \) mm. In trimming models, accurate and reliable flow stress models are considered as highly necessary to represent workpiece material constitutive behaviour, the constitutive material model reported by Johnson and Cook is employed in this study, it is often used for ductile materials in cases where strain rate vary over a large range and where adiabatic temperature increase due to plastic heating cause material softening.

Sample results of numerical and experimental investigations at plastic flow phase and cracking phase of the trimming process are shown in Figures 2-5.

In experimental studies, an advanced vision-based technology based on digital image correlation (DIC) for monitoring the cutting process is used (Fig. 4b). In two-dimensional digital image correlation, displacements of material are directly detected from digital images of the surface of an object. Then, the images on the surface of the object, one before and another after deformation are recorded, digitized and stored in a computer as digital images. These images are compared to detect displacements by searching a salient features from one image to another. Using this method it is possible to determine the areas of strong nonlinearities and deformation of material structure (Figs. 4 and 5).

References


Measurement aided computation of extensible cable deflections

Witold Cecot\textsuperscript{1}, Slawomir Milewski\textsuperscript{2}, Janusz Orkisz\textsuperscript{3,}\textsuperscript{*}
\textsuperscript{1,2}Civil Engineering Department, Cracow University of Technology
Warszawska 24, 31-155 Cracow, Poland
\textit{e-mail: plcecot@cyf-kr.edu.pl}\textsuperscript{1}, s.milewski@L5.pk.edu.pl\textsuperscript{2}

Abstract

The paper addresses computation of extensible cable deflection under static loads. The computation is aided with temperature and slope measured by a monitoring devise positioned at certain point of the cable. Large displacements in 3D are considered, including temperature influence as well as elastic supports. Various formulations and computational approaches are applied, namely the standard finite element method (FEM) as well as meshless finite difference method (MFDM). The MFDM is based on two different formulations of the considered problem (standard variational principle as well as the meshless local Petrov-Galerkin (MLPG) formulation). An appropriate approach for taking into account the measurement data is proposed. The paper is illustrated with a number of numerical examples.

Keywords: extensible cable, large displacements, finite element method, meshless methods, structure monitoring

1. Introduction

The problem of determination of a hanging chain or cable configuration is not new. One of the oldest models leads to the catenary curve which is the graph of the hyperbolic cosine function, similar to parabola. However, it allows for taking into account only the dead load (self-weight) and neglects cable extensibility. Therefore, a more complex model was used. It is based on the following assumptions: large displacements, small strains, negligible bending stiffness $EJ = 0$, finite tensile stiffness $EA < \infty$, no compressive stiffness, elastic constitutive relation with the effective Young modulus ($E$) as well as constant cross-section area $A$. Both dead and transient loads as well as temperature change due to various factors and the elastic supports are assumed. The above assumptions lead to a non-linear differential equations with appropriate boundary conditions. The solution is approximated by three computational approaches. The finite element method (FEM) as well as the meshless finite difference method (MFDM, [2]) are applied for Newton-Raphson solution as approximated by three computational approaches. Therefore, appropriate solution approach, taking into account both the theoretical assumptions as well as the experimental measurements, have to be carried out. Appropriate version of physically based approximation (PBA, [3]) is proposed, in which determination of the optimal balance between theory and measurements is done.

2. Formulation

The Lagrangian description is used to calculate extensible cable displacements [1]. The parametric equations of its static configuration in a Cartesian coordinate system are assumed in the following form

\begin{equation}
\begin{aligned}
x_1 &= X + u_1(X) \\
x_2 &= u_2(X) \\
x_3 &= u_3(X)
\end{aligned}
\end{equation}

where $X$ is the Lagrangian coordinate in the initial undeformed configuration, $X \in [0, L]$. $L$ is a distance between poles. The displacement vector components $u = [u_1(X), u_2(X), u_3(X)]$ are defined by the following system of ordinary differential equations

\begin{equation}
\frac{\partial F_i}{\partial X} = p_i, \quad F_i = AE\varepsilon - \alpha \Delta T \frac{\partial u_i}{\partial X}, \quad i = 1, 2, 3
\end{equation}

\begin{equation}
\varepsilon = (1 + (u_1')^2) + (u_2')^2 + (u_3')^2 - 1
\end{equation}

\begin{equation}
FN + K^{(s)}(u - \bar{u}) = 0, \quad \text{for} \quad X = 0, L
\end{equation}

where $\alpha$ is the effective thermal expansion coefficient, $\Delta T$ is the temperature increment, $\bar{u}$ is the positions of the unloaded supports, $N$ denotes the outward cross section unit normal vector in the undeformed configuration ($N = \pm 1$), $F_i$ are the axial force components, $\varepsilon$ is the axial strain, $K^{(s)}$ is a matrix of elastic support parameters at $X_1 = 0$ and $X_2 = L$,

\begin{equation}
p_i = D_{ij} q_j - g_i
\end{equation}

\begin{equation}
D = \text{diag}(\sqrt{(u_1')^2 + (u_2')^2}, \sqrt{(1 + u_1')^2 + (u_3')^2}, \sqrt{(1 + u_1')^2 + (u_2')^2})
\end{equation}

\textsuperscript{*}This research was supported by the National Center for Research and Development (NCBiR) and The National Fund for Environmental Protection and Water Management (NFOSiGW) under grant NCBiR/214108
where $\lambda \in [0, 1]$, satisfying the equality $A(u) = f$ and inequality constraints $B(u) < \Delta$, where $\Phi^T$ and $\Phi^E$ are non-dimensional theoretical and experimental parts of the functional $\Phi$, weighted by parameter $\lambda$. Several formulations of such optimization problem are considered, based on the experimentally measured quantities ($\Phi^E$), including cable slope, temperature, wind and theoretical rules of mechanics ($\Phi^T$), like the variational principle (6). In this manner, we may find enhanced values of searched quantities, especially more reliable cable configuration.

5. Numerical example

Proposed solution approaches (FEM, MFDM, MLPG5/MFDM) were verified on a benchmark problem presented in [1] and compared on the example with the following data: $L = 304.8m, A = 5.48e-4m^2, E = 13.10e5N/m^2, \Delta T = 20^\circ C, \alpha = 20e-6/(m/\circ C), q_1 = q_2 = q_3 = -20e-3N/m, \mu = 1e-2N/m$ and the poles heights difference of 5m. Regular mesh with 21 nodes was assumed. Results are shown in Fig. 1. They show good agreement.

![Figure 1: Plot of a cable deflection for three approaches](image)

Using one of the simplest PBA formulations, a simple example was taken. We solved first the theoretical problem (6) for $T^E=20^\circ C$ and undeformed cable length $L = 312.73m$. We found from there the cable slope tangent at $X=40m$, equal to $-0.0745$. Equivalent simulated “experimental” value was assumed as $-0.0845$. Applying the PBA enhancement procedure at $\lambda=0.8$, and tolerances assumed as $0.5^\circ C$ for $\Delta T$, $0.02$ for slope, and $1m$ for $L$, we solved the constrained optimization problem, obtaining finally $T^E=20.5^\circ C, L=312.00m$ and slope $-0.0764$. These enhanced values are the compromise between the theoretical prediction and experimental data.

References


Accuracy of Lattice Boltzmann Method in application to multiphase tribological flows

Michał Dzikowski¹, Jacek Rokicki²

¹,² Institute of Aeronautics and Applied Mechanics, Warsaw University of Technology
Nowowiejska 24, 00-665 Warsaw, Poland
e-mail: mdzikowski@meil.pw.edu.pl

Abstract

Detailed analysis of the cavitating flow properties inside the tribological devices is needed in development and optimisation process of those devices. Lattice Boltzmann method is a potential choice for numerical tool for such flows. In the work, diffuse interface method LBM for multiphase flows is evaluated in terms of accuracy and applicability to bubbly flow in narrow channels and gaps. As in every discrete interface method, in LBM phase change occurs on few grid cells, therefore influence of this thickness on simulation results was also analysed. Comparison is presented of obtained results with experimental data, together with grid independence study.

Keywords: Lattice Boltzmann Method, multiphase flow, grid dependence

1. Introduction

The Lattice Boltzmann method (LBM) is a well-established numerical approach, widely used for numerical simulations of single phase fluid flows. It is particularly popular in areas, where extremely complex geometry (e.g., for porous media flows) prevents from application of more traditional methods based on finite element or finite volume discretisations. The LBM is also used successfully for multiphase flows, even though many unresolved issues appear in this case (the presence of spurious velocity field, lack of in/out flow boundary conditions, etc. [4, 6, 10, 12]). In this study, we investigate the multiphase Lattice Boltzmann scheme reported by [7]. This scheme is used to simulate a long vapour bubble rising in a narrow channel filled with liquid and bubble trapped in cavity.

2. Diffuse interface method in LBM

Multiphase diffuse interface models are based on momentum equation for van der Walls fluid (single component, multiphase fluid with surface tension considered) [5, 13]:

\[
\frac{d\psi}{dt} = \nabla \cdot \left( (-P^0(\rho, T) + \frac{\lambda}{2}(\nabla \rho)^2 + \lambda \rho \nabla^2 \rho) \mathbf{I} + \lambda \nabla \rho \otimes \nabla \rho \right) + \mathbf{F}_{MP} \quad (1)
\]

\[
\frac{d\rho}{dt} = -\rho \nabla \cdot \mathbf{v} \quad (2)
\]

where \( \mathbf{v} \) and \( \rho \) are fluid velocity and density respectively. A symbol \( P \) stands for pressure, and \( P_0(\rho, T) \) is a non-ideal equation of state (EOS). Surface tension is controlled by parameter \( \lambda \). In the presented approach, non-ideal part of momentum equation (non-ideal EOS and surface tension) were introduced to standard scheme by means of internal force \( \mathbf{F}_{MP} \). As there is no direct interface tracking, those methods are straightforward to implement. Main drawback comes from the fact, that in numerical simulation we need to discrete interface with enough grid points to recover gradients of macroscopic values, mainly density. In most cases however, interface has few free paths thickness so it is not practical to have so much dense grid in simulation. In most simulations we drop strict restriction on some parameters, and preserve others, to obtain reasonable interface thickness. Basic idea is that we smear interface on few cells, but parameters like surface tension or heat coefficients are preserved. This method is referred to as second gradient method after [5].

In presented cases, additional term \( \mathbf{F}_{MP} \) was discretised as [7]:

\[
\mathbf{F}_{MP} = 2\psi \nabla \psi = \frac{1}{\alpha h} A \sum_{k=1}^{N} G_k \psi^2(x + \mathbf{e}_k) \mathbf{e}_k + \frac{1}{\alpha h} (1 - 2A) \psi(x) \sum_{k=1}^{N} G_k \psi(x + \mathbf{e}_k) \mathbf{e}_k 
\]

(3)

where \( k \) = \( \frac{P_{0} \Delta t^{2}}{h^{2}} \), \( \psi(\hat{\rho}, \hat{T}) = \sqrt{-k \hat{P}(\hat{\rho}, \hat{T}) + \hat{\rho} \hat{\theta}} \).

3. Bretherton bubble flow

The proposed LB method is validated for a gravity driven flow of the bubble train (see Fig. 2). The channel remains periodic in order to avoid problems with inflow/outflow boundary conditions (for the authors best knowledge there is still lack of consistent boundary conditions, both for the inlet and the outlet). This configuration is known as Bretherton or Taylor bubble flow. It was a subject of many experimental [11], theoretical [1] and numerical studies [3, 9]. In the work, this case was used to test parameters of the method, grid dependency and stability. It was tested, that proposed scheme is a least first order in term of recovered film thickness. Also it remains stable in parameter range predicted by Kupershtock [8], but tested for fully periodic case without boundary by original author. The driving gravity force is denoted \( G \), while \( H \) and \( h_{\infty} \) denote the channel height and the equilibrium (in the flat region of the long bubble) film height respectively. The length-to-height ratio was 20, except the case of the convergence study, when it was raised up to 25.
4. Bubble stretching in a cavity flow

The second test problem was a periodic channel flow with step change in a channel height, with a bubble trapped in the gap. The geometry is presented in Fig. 3. Periodic boundary conditions are present on the inlet and the outlet. Coyne et al. [2] delivered system of ODEs, that describes shape of phase interface, which are similar to those provided by Bretherton for a long bubble in a tube [1]. The work of Coyne et al. [2] is commonly used in development of boundary condition for cavitating tribological flow. Authors found, that inclusion of periodicity into original problem leads to significantly different results. Unfortunately, the lack of proper pressure/velocity boundary conditions for LBM prevents from full comparison of both cases.

5. Conclusions

The main goal of the presented work was to evaluate accuracy of the method in tribological cases. Mesh dependence and accuracy of the method was analysed. The influence of interface thickness on simulation results was also evaluated. Two test cases were chosen: Bretherton bubble flow and the liquid dragout from a cavity. The obtained results were compared with some theoretical predictions as well as with available experimental data. Influence of periodicity and different boundary conditions should be investigated and are subject of further study.

References

A numerical scheme of shift-periodic boundary condition for LBM

Arkadiusz Grucelski¹, Jacek Pozorski²

¹,²Institute of Fluid-Flow Machinery
Fiszera 14, 80-231 Gdańsk, Poland
e-mail: agrucelski@imp.gda.pl, jp@imp.gda.pl

Abstract

With the development in computational techniques, detailed simulations of flow thermomechanics in granular media at the level of a single pore are attracting increasing interest. In numerical computations porous media are often regarded as a representative element of volume in multiscale modelling approach. The Lattice Boltzmann method (LBM) is more and more often chosen for modelling of fluid thermomechanics and other phenomena in complex geometries. In the work we present a novel numerical scheme of shift-periodic boundary condition for internal energy distribution function in LBM, along with recent results for the temperature profiles at the inlet/outlet with accompanying pressure profiles.

Keywords: fluid flow and heat transfer, Lattice-Boltzmann, periodicity condition

1. Introduction

Recently, there has been a growing interest in multiscale simulations. The main reason is the increasing expectation for results of, as much as possible, detailed simulations of complex phenomena. The analysis at many levels of scales is triggered by advances in computer power and developments of new computational methods. All this allows for detailed simulations of various phenomena at the pore level of representative element of volume (REV). The physico-chemical and geometrical complexities imply that more traditional tools and software of computational fluid dynamics (CFD) sometimes reveal to be prohibitively expensive as far as detailed modelling is concerned. Therefore, our longer-term idea is to develop a multiscale approach with a microscopic (single-pore level) 3D/2D computation in the REV domain, followed by a macroscopic, physically-sound analysis of the process in terms of a 1D/2D model. Multiscale modelling, with wide perspectives for fluid dynamics, is a demanding approach. One needs to provide detailed description of geometry (at the REV level and also at level below) and numerical tools together with boundary schemes prepared for physically sound inlet/outlet boundary conditions.

One of the numerical tools (operating at meso-scale level) recently gaining attention of the scientific and industrial communities is the Lattice Boltzmann method (LBM). The method has proved suitable for simulation of viscous and nearly incompressible flows in simple and complex geometry, as well as heat transfer. Having already attracted interest as a tool of computational fluid dynamics, LBM is more and more used for modelling fluid-structure interactions, chemical reactions and species transport, non-Newtonian flows, turbulence, etc.

As a first development step towards the physically-sound description of the coking process, the authors applied the LBM to simulate fluid flow past a cylinder and in simple granular (or porous) media [2]. In the second step, we have dealt with non-isothermal flow in a simple and complex geometry [3]. The present work addresses an open problem of correct formulation and implementation of shift-periodic conditions at the inlet and outlet boundaries of the REV domain (the bed of grains, like the coal grains during the coking process). In case of fluid flow, some solutions were introduced and next verified by Zhang and Kwok in [5]. To the best knowledge of the authors, the proposal of a scheme for periodic condition with a given temperature difference between inlet/outlet for heat transfer simulations in LBM, is presented here for the first time.

2. Numerical modelling

2.1. Lattice Boltzmann method

In the present work we use the Lattice Boltzmann method, where the flow density and velocity are solved in terms of the density distribution function (usually denoted by f) as it is presented in [2]. The temperature field is found from the internal energy density distribution function (usually denoted by g), see [3] and references therein. The form of all LB equations used here is similar and the D2Q9 discretization scheme for f, g advection is used: f(x, t, v) → f_i(x, t). The macroscopic flow density ρ, velocity and temperature θ are found from suitable averaging [4].

2.2. Boundary schemes

At the solid/fluid interfaces, we use the non-equilibrium boundary scheme for fluid flow and heat transfer: the no-slip condition is applied for velocity and a known temperature is assumed at the surface of uniformly heated obstacles.

At the longer side of computational domain (parallel to flow direction) we use the standard periodic condition. At the boundaries perpendicular to the main flow direction in case of fluid flow we use the shift-periodic boundary conditions (with pressure drop between inlet and outlet calculated from the Darcy law). Briefly, shift-periodic conditions at the flow inlet and outlet are, see [5]:

\[ f_{\text{inlet}} = f_{\text{outlet}} \left( \frac{\rho_{\text{inlet}}}{\rho_{\text{outlet}}} \right) + \beta/c_s^2, \quad f_{\text{inlet}} = f_{\text{outlet}} - \beta/c_s^2 \left( \frac{\rho_{\text{inlet}}}{\rho_{\text{outlet}}} \right) \]

where the superscripts describe nodes (inlet, outlet) and the subscripts identify the advection direction of population (where e_{out} points outside the numerical domain, for leaving populations), see Fig. 1. The operation (\langle \rangle) is the averaging over inlet or outlet nodes and \( \beta \) is a constant representing the overall pressure gradient; \( \rho_0 \) is the initial density and \( c_s \) is the speed of sound in LBM.
The main idea of the novel boundary scheme for $g$ is:

\[
\begin{align*}
\varrho_{\text{inlet}} & = \varrho_{\text{outlet}} \theta_{\text{ref}} / \langle \varrho_{\text{outlet}} \rangle - \Omega_1 \Delta \theta \\
\varrho_{\text{outlet}} & = \varrho_{\text{inlet}} \theta_{\text{ref}} / \langle \varrho_{\text{inlet}} \rangle + \Omega_1 \Delta \theta,
\end{align*}
\]

respectively for the inlet and the outlet. Here $\Delta \theta$ represents the temperature difference between inlet and outlet, $\Omega_1$ are weights. In our earlier work [3] a modified scheme was presented to set proper shift-periodic condition in the geometry (Fig. 2) and $\Delta \theta$ was computed, based on [1].

3. Benchmark flow case

Numerical modelling was performed in a periodic geometry of staggered square rods, see Fig. 2. Detailed description of the geometry is presented in [1]. The flow domain is discretized with use of $N \times 2N$ nodes, where $N = 100$. The obstacles are identical squares of size $d = 0.4N$, resulting in the porosity value of $\varepsilon = 0.84$.

At the beginning of simulation, we set the temperature to $\theta_0$ in the whole numerical domain. Then, the solid obstacles are uniformly heated to $\theta_0(x)$. For described test case, the obstacles’ temperatures are kept at $\theta_0(x = N/2) = \theta_0 + 0.25 \Delta \theta$ and $\theta_0(x = N) = \theta_0 + 0.75 \Delta \theta$, for the first and second row of rods, respectively (see Fig. 2). After an initial transient, the proposed shift-periodic condition was applied for heat transfer with the imposed temperature difference between the inlet and the outlet $\Delta \theta = 0.25 \theta_0$. For this work, using the geometrical symmetry, we choose $\theta_{\text{ref}} = \langle \theta \rangle(x = N/2)$. The pressure difference between inlet and outlet is calculated with use of the Darcy equation with permeability calculated by the Kozeny-Carman correlation.

Figure 2: The periodic geometry used for benchmark simulation of convective heat transfer. The flow is from left to right, the pressure map is in grayscale; the isolines of temperature are seen.

Figure 3: a) The temperature profiles (averaged over $y$ direction) along the REV domain: at the beginning of simulation ($t_{\text{init}}$), at an intermediate time ($t_1$) and at the steady state ($t_{\infty}$); b) the steady-state temperature and pressure profiles at inlet and outlet.

4. Future work

A test case similar to the case described in [5], with the geometry presented here (Fig. 2) for the temperature field is under preparation. Another computation will consider a granular medium with a uniform (yet random) arrangement of grains. We aim to compare the temperature profiles at inlet/outlet of every period. A next-term target is 3D simulation. A full set of results will be presented at the Conference.

References


Coupling of Finite Element Method and meshless Finite Difference Method with nonconforming approximation orders

Jan Jaskowiec¹, Sławomir Milewski²

¹,²Institute for Computational Civil Engineering Faculty of Civil Engineering, Cracow University of Technology, ul. Warszawska 24, 31-155 Kraków, Poland

e-mail: j.jaskowiec@L5.pk.edu.pl¹, s.milewski@L5.pk.edu.pl²

Abstract

The paper presents the coupling technique for the finite element method (FEM) and the meshless finite difference method (MFDM), applied for different subdomains of the same domain of a boundary value problem. In contrast to other coupling techniques, it is possible here for the FEM and MFDM to have different discretization densities and approximation orders (e.g. first for the FEM, higher order for the MFDM). In spite of the usage of different orders for the FEM and MFDM, the final solution does not exhibit a discontinuity along the common boundary. The proposed approach is illustrated with a two-dimensional benchmark example.

Keywords: MFDM, FEM, coupling

1. Introduction

The paper presents the effective technique for coupling of two computational approaches, namely the finite element method (FEM) and the meshless finite difference method (MFDM). This work may be considered the extension of the paper [2]. The coupling technique, in contrast to other coupling techniques, allows to coincide FEM and MFDM subdomains with nonconforming discretisations as well as approximations orders. Moreover, no transient elements are required, which were applied in e.g. [3, 1].

The appropriate integral along the inter-subdomain border has to be calculated instead. It is based on the values of shape functions on both sides of the border. It may be stated that the inter-subdomain integral enforces continuity of the approximation in spite of the fact that discretisations and approximation orders on the both subdomains are completely different. In this paper, FEM is based on Bubnow-Galerkin formulation while MFDM is constructed using Petrov-Galerkin approach.

The approach presented in this paper is illustrated with two dimensional benchmark example. The exact solution has exponential form and is characterised by rather large gradients. The domain is divided in such a manner that the part of the domain with large gradients is covered with nodes only (for the MFDM) with higher order approximation.

2. The FEM–MFDM coupling technique

2.1. The problem formulation

For the sake of clarity, the paper discusses solely the stationary heat transport problem. The thermal model is based on the commonly applied strong form of energy balance with appropriate boundary conditions of essential and natural types

\[ \text{div} \mathbf{q} = r, \quad \mathbf{q} = -\lambda \nabla \Theta \quad \text{in} \ V \]

\[ \Theta = \bar{\Theta} \text{ on } S_{\Theta}, \quad \mathbf{q} \cdot \mathbf{n} = \bar{h} \text{ on } S_h \]

\[ \Theta \] is the temperature field, \( \mathbf{q} \) is the heat flux vector, \( r \) is the heat source intensity, and \( \lambda \) is the thermal conductivity parameter.

The considered problem is defined on domain \( V \) and its outer boundary \( S \). The domain \( V \) is divided into two subdomains: \( V^h \) with the finite element discretization and approximation, and \( V^m \) to which the meshless method is applied. Those two subdomains do not overlap on each other but they have the common inter-subdomain border \( S_d \):

\[ V^h \cup V^m = V, \quad V^h \cap V^m = S_d \]

As the consequence, both the finite element nodes and the meshless nodes may be situated on \( S_d \), though in different locations.

2.2. Meshless finite difference shape functions

The generation of the shape functions for the MFDM is based on the moving weighted least squares (MWLS) approximation which is spanned over nodes belonging to the finite difference stars (stencils) [4]. They can be defined as follows

\[ \Phi_m = p^T (0)(P^T WP)^{-1} P^T W \]

where \( p \) is a vector of basis monomial functions taken from the Taylor expansion, \( W = \text{diag}(\omega_i), i = 1, \ldots, m = \text{diagonal weights matrix, with } \omega_i = \omega(||\mathbf{x} - \mathbf{x}_i||) \), matrix \( P \) is defined as

\[ P = \begin{bmatrix} 1 & h_1 & k_1 & \frac{1}{2} h_1^2 & \ldots & \frac{1}{p} h_1^p \\ 1 & h_2 & k_2 & \frac{1}{2} h_2^2 & \ldots & \frac{1}{p} h_2^p \\ & \vdots & \vdots & \vdots & \ddots & \vdots \\ 1 & h_m & k_m & \frac{1}{2} h_m^2 & \ldots & \frac{1}{p} h_m^p \end{bmatrix} \]

with \( h_i = x - x_i, \quad k_i = y - y_i, (x, y) \) is the evaluation point.

2.3. The weak formulation

The approximations over \( V^h \) and \( V^m \) are constructed independently, which leads to the discontinuity in approximation over \( S_d \). In spite of this discontinuity, the degrees of freedom are set to enforce continuity in the final solution.

It is assumed that along \( S_d \) a thin amount of material of \( \varepsilon \) width exists. The same assumption holds along \( S_{\Theta} \), but with \( \frac{1}{2} \varepsilon \) thickness which allows for enforcing essential boundary conditions. Under such assumptions, the weak formulation of the
boundary problem (1) leads to the following equation
\[
2 \int_{S_{0}} \frac{\lambda}{2} \nu \Theta \, dS + \int_{V} \lambda \nabla \nu \cdot \nabla \Theta \, dV + \int_{S_{d}} \frac{\lambda}{2} [v][\Theta] \, dS + \\
\int_{S_{d}} \lambda \langle \nabla \nu, Q_{v}, Q_{\Theta} \rangle \, dS = \\
2 \int_{S_{0}} \frac{\lambda}{2} \nu \Theta \, dS + \int_{V} v \nu \, dV + \int_{S_{d}} z \, r \, dS
\]  
(6)
where \( Q_{v} = I - n^{d} \otimes n^{d} \). \( n^{d} \) is unit vector normal to \( S_{d} \), the operators \([ ]\) and \( \langle \rangle\) for a function \( f \) are defined as follows:
\[
[f](x) = f(x + z n^{d}) - f(x - z n^{d})
\]  
\[
\langle f \rangle(x) = \frac{1}{2} \left( f(x + z n^{d}) + f(x - z n^{d}) \right)
\]  
(7)

2.4. Discretization

In the proposed approach, completely different discretization and approximation schemes are applied for \( V^{h} \) and \( V^{m} \) subdomains. It leads to a separate set of shape functions as well as degrees of freedom. Therefore, the approximation of temperature \( \Theta \) is expressed in the following manner
\[
\Theta = [\Phi_{h}, \Phi_{m}] [\Theta]_{h} \Phi_{h} = \Phi \Theta
\]  
(8)
where \( \Phi_{h}, \Phi_{m} \) are the matrices of shape functions for \( V^{h} \) and \( V^{m} \), respectively; \( \Theta_{h} \) and \( \Theta_{m} \) are the vectors of degrees of freedom for \( V^{h} \) and \( V^{m} \), respectively; \( \Phi \) is a matrix that consists of all shape functions, and \( \Theta \) is the vector with all degrees of freedom.

Providing that normal \( n^{d} \) is directed toward \( V^{h} \), the matrices \( [\Phi] \) and \( \langle \Phi \rangle \) are described on \( S_{d} \) as
\[
[\Phi] = [\Phi_{h} - \Phi_{m}] , \quad \langle \Phi \rangle = 0.5 \left[ \Phi_{h} \Phi_{m} \right]
\]  
(9)
The approximation formulas (8) and (9) are applied to test function \( v \) as well. After their substitution to (6) one obtains the following algebraic system of equations.

3. Example

The proposed approach is illustrated with two-dimensional benchmark example on a square domain \([-1, 1] \times [-1, 1]\). The exact solution of the example is
\[
\Theta(x, y) = \exp \left( -5 \cdot \left( x^{2} + \left( y - \frac{9}{10} \cdot x \right)^{2} \right) \right)
\]  
(10)
The function constitutes the exact analytical solution to the boundary value problem
\[
\begin{align*}
-\Delta \Theta &= f(x, y) \quad \text{in} \ V \\
\Theta &= \Theta_{0} \quad \text{on} \ S_{e}
\end{align*}
\]  
(11)
in which the Dirichlet boundary conditions are applied for all parts of the outer boundary.

The problem defined in (11) may be considered a heat flow conductivity problem in which the thermal conductivity \( \lambda = 1 \), and the heat source \( r \) is given by the \( f(x, y) \) function.

The considered domain is divided onto \( V^{h} \) and \( V^{m} \) subdomains in such a manner that \( V^{m} \) covers the region with large gradients. In the \( V^{h} \) linear finite elements are applied, while in \( V^{m} \), the approximation of \( p = 3 \) order is assumed.

Figure 1 presents results for the considered example in the form of the mixed FE-MFD discretization (with nodes \( n = 512 \) and elements/integration cells \( N = 910 \)) and the graph of the numerical solution.

4. Conclusions

In the paper, the FEM and MFDM are coupled in the same domain of the boundary value problem. In the coupling technique, different approximation orders as well as discretization densities for both methods may be applied. It is based here on the additional integral defined on the common inter-subdomain border with finite (but very small) width. Preliminary results are encouraging. Development of the relevant adaptation techniques as well as analysis of the coupled thermomechanical problems are planned.

References

On the application of multipoint meshless method to the nonlinear analysis

Irena Jaworska¹, Janusz Orkisz²
¹,² Institute for Computational Civil Engineering, Cracow University of Technology
ul.Warszawska 24, Cracow, 31-155 Poland
e-mail: irena@L5.pk.edu.pl ¹, plorkisz@cyf-kr.edu.pl ²

Abstract

A new multipoint meshless finite difference method using higher order approximation technique is discussed here. It is based on arbitrary irregular meshes, the MWLS approximation and the local or various global formulations of boundary value problems. Following L.Collatz [1] the general and the specific versions of the multipoint MFDM are distinguished. The general multipoint approach can be used for all types of b.v. problems including the nonlinear ones. The paper is focused on the possibility of application of this method to nonlinear analysis. The numerical results obtained so far for several simple problems are very encouraging.

Keywords: meshless finite difference method, higher order approximation, multipoint method, nonlinear analysis

1. Introduction

The higher order multipoint meshless finite difference method (MFDM) [4,6] has recently been developed by the authors for analysis of boundary values (b.v.) problems. The concept of the multipoint approach is based on raising the approximation order of searched unknown function by using a combination of its values together with a combination of additional degrees of freedom at all finite difference (FD) star nodes. The known values of considered equation right hand side (specific case) or function unknown derivatives (general case) may be used as the additional d.o.f. This improves the FD solution without increasing the number of nodes in the mesh.

The method formulation, following the original Collatz [1] multipoint FD concept, has been modified and extended to the multipoint MFDM [4]. For this purpose the multipoint method is based on the moving weighted least squares (MWLS) approximation technique [2] instead of the polynomial interpolation, as proposed by Collatz. Moreover, rather than regular meshes, unstructured, totally irregular clouds of nodes [3], as well as local or various global formulation of b.v. problem may be applied here.

Two basic versions of the multipoint MFDM are considered: the general and the specific ones [6]. Application of the specific approach is simpler and easier in implementation, but is mainly restricted to linear b.v. problems. The general formulation though more complex, may be used for all types of b.v. problems including nonlinear ones. In each of these multipoint MFDM cases, higher order approximation may be obtained, using the same FD star, as needed to generate FD operators in the classical MFDM approach.

Besides the improved solution quality, the general multipoint case potentially shows a wide range of applications. The reason is that using the multipoint MFDM, all derivatives up to p order may be defined from relations \( u^{(p)} = u \) to searched function without additional effort. In this case the relations \( u^{(p)} = u \) depend on discretization of the problem domain only. Therefore, the solution of almost all problems in computational mechanics, including nonlinear ones may be obtained this way.

This paper presents the first attempt of application the multipoint meshless FDM to the nonlinear analysis.

2. Multipoint general approach formulation

Consider the local formulations of boundary value problems for the n-th order PDE

\[
\begin{align*}
\mathcal{L}u &= f, & u(P) &= \mathcal{P}u, & \text{for } P \in \Omega \\
\mathcal{G}u &= g, & \partial u(P) &= \mathcal{P}u, & \text{for } P \in \partial \Omega
\end{align*}
\]

or an equivalent global formulation involving integral.

In the multipoint formulation, the meshless FDM difference operator \( \mathcal{L}u \) is obtained by the Taylor series expansion of an unknown function \( u \), including higher order derivatives, and using additional degrees of freedom at nodes. For this purpose may be applied e.g. combination of the right hand side values \( f_i \) of the considered differential equation at each node of MFD star:

\[
\mathcal{L}u \approx \mathcal{L}u_0 = \sum_j c_j u_j = \sum_{k} c_{k} f_j \Rightarrow \mathcal{L}u = Mf_j.
\]

This is the basic formula for the multipoint specific formulation. Here \( f_j \) is the number of a node in a considered MFD star, \( Mf_j \) is a combination of the equation right hand side values \( f_i \) may present value of the whole operator \( \mathcal{L}u \) or its derivatives.

In the general multipoint method, a specific derivative \( u^{(k)} \) is used as additional d.o.f. instead of the right hand side of the given differential equation

\[
\sum_j c_j u_j = \sum_{k} \alpha_{k} u^{(k)}
\]

Application of the specific case is restricted to the linear b.v. problems. When the specific formulation cannot be applied (e.g. for nonlinear b.v. problems), the various versions [5] of the general one may be used.

In the multipoint MFDM general discretization concept, the partial problems providing relations between searched function \( u \) and its derivative \( u^{(k)} \) in the whole domain \( \Omega \) should be solved first. Function \( u_j \) and derivative \( u^{(k)} \) are developed into the truncated Taylor series. Afterwards the weighted error functional

Błąd! Nie można tworzyć obiektów przez edycję kodów pól.
is generated. Minimization of the functional \( \tilde{J} \tilde{\varv}(D u) = 0 \) yields at each node \( i \) the local multipoint MFD formulas for \( q = 0,1,\ldots,p \) order derivatives \( D_{u}^{q} \), and finally for the basic equation \( \sum_{c} c_{i} u_{i} = \sum_{c} \alpha_{c} u_{i}^{(k)} \). Simultaneous equations \( \hat{A} u = \hat{B} u^{(k)} \) are generated in the whole domain \( \Omega \). The solution provides for each \( k = 1,\ldots,n \) the relation \( u^{(k)} = \hat{C} u \).

It is also possible to introduce an alternative approach, in which only the first order relation \( u = \hat{C} u \) is generated. This formula is subsequently differentiated and used to find the derivative of the required order \( u^{(k)} = (\hat{C})^{k} u \). In the \( N \)-dimensional space the matrix \( \hat{C} \) has to be computed only \( N \) times, independently of the derivative order required.

Having found the FD relations \( u^{(k)} = u \) in the whole domain \( \Omega \) for all derivatives emerging in the considered b.v. problem, these relations are applied to a given nonlinear problem. After such discretization PDE depends on the primary unknowns \( u \) only. Collocation approach, carried out in the whole domain \( \Omega \cup \partial \Omega \) may be used then in order to generate the simultaneous FD equations and provide the final solution.

The same multipoint MFD approach may be applied in the case of the global or local-formulations [7] of the b.v. problem. In the local one, the integral is discretized using the formulas for derivatives found in the way mentioned above.

### 3. Numerical results

Preliminary tests of application of the new multipoint method into nonlinear analysis were carried out. Among other, the nonlinear engineering problem – deflection of the ideal membrane has been solved this way. The differential equation

\[
\left\{ \begin{array}{l}
\dot{u}_{x}^{x} + \dot{u}_{y}^{y} + \left( u_{x}^{x} \right)^{2} + \left( u_{y}^{y} \right)^{2} - 2 \left( u_{x}^{x} \right)^{2} = \rho \\
\left( \dot{u}_{x}^{x} + \dot{u}_{y}^{y} \right)^{2} - 2 \left( u_{x}^{x} \right)^{2} = \rho
\end{array} \right. \quad \text{in} \quad \Omega
\]

and the appropriate boundary conditions were given where \( u = u(x, y) \) is the membrane deflection, problem domain \( \Omega \) – is the membrane middle surface before deformation, and pressure \( \rho \) – is the external load.

The total load \( P \) was divided into several load increments. For each one magnitude the Newton-Raphson iterative approach was applied. The tangent matrix was found using symbolic differentiation, whereas partial derivatives were numerically evaluated using appropriate multipoint MFD formulas described in the previous section.

The problem was examined in the circular domain shape, with radius \( r = 2m \) and load \( \rho = 1 \text{kN/m}^2 \). Moreover, the irregular cloud of nodes was applied (Fig. 1).

![Image](image-url)

**Figure 1:** Irregular random mesh of circular domain and numerical solution for ideal membrane deflection

The solution convergence of Newton-Raphson method with 3rd order general multipoint approach applied is presented in Fig. 2.

### 4. Final remarks

The paper presents the first attempt to apply the multipoint meshless FDM in the analysis of nonlinear problems. Two basic cases of multipoint method were considered namely the general and specific ones. The general case is relatively complex, but it allows for analysis of all types of b.v. problems, including nonlinear ones. The proposed approach is expected to work well especially in the case of geometrical nonlinearity.

Preliminary tests of application of the general multipoint method in nonlinear analysis were carried out. The numerical results obtained so far for several simple problems are very encouraging, taking into account both their precision and efficiency.

The approach is under current development with plans for a further research.

### References


Flow patterns generated by a flapping airfoil

Tomasz Kozłowski¹, Henryk Kudela¹

¹,²Faculty of Mechanical and Power Engineering, Wroclaw University of Technology
Wybrzeże Wyspianskiego 370, Wroclaw, Poland

e-mail: tomasz.kozlowski@pwr.wroc.pl¹, henryk.kudela@pwr.wroc.pl²

Abstract

Flapping is the basic motion for birds insects and fish. Fish, birds and insects, flap their wings or fins to an extremely effective movement in the surrounding fluid. It is well known that the aerodynamic forced generation in the flapping motion is ruled by the unsteady fluid phenomena. In the paper we present the unsteady effects led to the lift and thrust force generation of the flapping foil, for low Reynolds number regime, below, 10⁴ [3]. The earlier studies and observations of natural flyers brought the great contribution to understanding the wing kinematics and phenomena dominated by the vortex creations and interactions. The Vortex Particle Method is well suited to study the phenomena dominated by unsteady effect caused by vorticity dynamics. In the paper we demonstrate the unsteady effects that lead to the lift and thrust force generation of the flapping foil, for low Reynolds number flows. In order to study the dynamics of evolution of the vorticity the Vortex-In-Cell method was used. We limited our research to study the flapping elliptical wing in two dimensions.

Keywords: flapping motion, frree flight, vortex street, thrust, lift, Vortex-In-Cell method

1. Introduction

The earlier studies and observations of natural flyers brought the great contribution to understanding the wing kinematics and flow phenomena generated in flapping flight. In contrast to classical airplanes, most of the insect and birds operate in low Reynolds number regime, below, 10⁴ [3]. The simplified potential flow theories are not applicable for studying such a flight where the flow is dominated by unsteady effect caused by vorticity dynamics. In the paper we demonstrate the unsteady effects that lead to the lift and propulsion production for the two dimensional flapping profile at Reynolds number Re < 500. Although flight in nature is naturally three-dimensional the two-dimensional simplification is widely used and seems to be appropriate to capture the essence of flapping flight aerodynamics [4]. In order to study moving objects in a fluid we choose vortex particle method – Vortex-in-Cell method. The VIC method is well suited to study the phenomena dominated by the vortex creations and interactions. The method was carefully tested for problems with analytical solution and problems with well documented experimental data.

2. The Vortex Particle Method

The Navier-Stokes equations of the incompressible viscous fluid in primitive variables (u, p), can be transformed to the Helmholtz equations in vorticity - stream function formulation (ψ, ω). The equations of the vorticity evolution for two dimensional flows has the form

\[ \frac{\partial \omega}{\partial t} + (\nabla \omega) \cdot \mathbf{u} = \nu \Delta \omega, \]  
\[ \Delta \psi = -\omega, \]  
\[ \mathbf{u} = \nabla \times (0, 0, \psi), \]  

where \( \mathbf{u} = (u, v) \) is velocity vector. In the vortex particle method the continuous vorticity field was approximated by the discrete particles distribution:

\[ \omega(x, y) = \sum_p \Gamma_p \delta(x - x_p)\delta(y - y_p), \]  

where \( \Gamma_p = \int_A \omega(x, y) dx dy, A = h^2 \). For solution of the vorticity transport equation the viscous splitting algorithm is used. First, the inviscid fluid motion equation was solved

\[ \frac{\partial \omega}{\partial t} + \mathbf{u} \cdot (\nabla \omega) = 0. \]  

The Helmholtz theorems yield that vorticity is constant along the trajectory and is carry by the fluid particles. The motion vortices can be escribed by the finite set of ordinary equations

\[ \frac{dx}{dt} = u, \quad \frac{dy}{dt} = v, \quad x(0, \alpha_1) = \alpha_1, \quad y(0, \alpha_2) = \alpha_2, \]  

where the velocity field \( \mathbf{u} \) is obtained by the solution of the Poisson equation for the stream function and vorticity (2) using the numerical grid and (3). In the second step viscosity is determined by the solution of the diffusion equation

\[ \frac{\partial \omega}{\partial t} = \nu \Delta \omega, \]  
\[ \omega(x, y, 0) = \omega_0, \quad \omega|_{wall} = \omega_s, \]  

with the proper boundary conditions for vorticity at the wall \( \omega_s \). The vortex particles generate the non-zero tangent velocity on the wall. The nullifying of the vorticity on the wall is realized by introducing a proper portion of vorticity that ensure the \( \mathbf{u} \cdot s^V = 0 \) were \( s^V \) is unit tangential vector [1]. In order to solve the Poisson equation on the mesh, in each time step, the redistribution of vortex particle must be done onto the neighboring grid nodes using the interpolation formula that conserve three first moments.

In order to fit the numerical grid to the solid body we transformed the non-rectangular physical region \( x, y \)–variables to the rectangular one \( (\xi, \eta) \) by the conformal mapping. By the conformal mapping the non-regular flow region is replaced by the regular one, where the fast direct method for solution of the Poisson equation can be used.

3. Flow simulation and flapping flight

One of the most popular 2D flow cause by the flapping is called as the incline hovering [3]. It may be described by the equations [4]

\[ A(t) = \frac{A_0}{2} \cos(2\pi ft), \]  
\[ \alpha(t) = \alpha_0 + \alpha_m \sin(2\pi ft), \]  

where \( A(t) \) is the roof amplitude and \( \alpha(t) \) is the incline angle of the roof.

175
where \( x(t) \) denote instantaneous position of the wing center, \( A_0 \) is the plunging amplitude, \( f \) is the frequency, \( \alpha(t) \) is the angle of attack measured relative to the horizontal, \( \alpha_0 \) is the initial angle of attack and \( \omega_0 \) is the pitching amplitude. The translation and angular velocities are given by \( U_0 = \frac{dx}{dt} \) and \( \Omega_0 = \frac{d\omega}{dt} \). The inclination of the stroke plane was \( \beta = 60^\circ \). After first three strokes the vorticity around the profile is generated periodically. In figure 1 the streamlines and vorticity after the eighth stroke is presented.

Figure 1: Vorticity and stream lines for the back-stroke in the eighth stroke of the flapping wing, \( Re = 125 \). The stream lines are plotted for fixed to the wing coordinates; \( k \) means the reduced frequency [2].

Figure 2: The wing wake observed in flapping motion. On the left the vorticity field is presented, the time averaged velocity profile is also depicted. On the right the visualization with passive markers is presented.

In order to capture the essence in the vortex wing wake transitions, we reduced the flapping motion of the foil to the simple harmonic oscillation, that were perpendicular to the direction of main flow. In literature such foil motion is called plunging. We established the relationship between Strouhal number, amplitude of oscillations and vortices distribution in the vortex street behind the flapping foil. With a properly chosen amplitude of the oscillation observation can be made either of the reversed Karman vortex street producing the thrust force (the first two frames in figure 2) or the reversed and deflected vortex street which generates both thrust and positive lift force.

In order to check how the vorticity around the profile affects the dynamics and trajectory of the flight we simulated also the free flight. We assumed that the calculated hydrodynamic forces and the gravity force act through the center of gravity of the profile and its motion is decried according to second Newton law

\[
m [\ddot{x}, \ddot{y}] = [0, -g] + F_{fluid} + m \frac{dv_0}{dt}.
\]

where the velocity \( v_0 \) was calculated from the equation (9). The vorticity around the free flying profile was presented on figure 3. The stable flight was possible only when the distribution of the vortices around the profile has well organized periodic structure.

Figure 3: Vorticity generation in free flapping flight. By a small modification of the \( \alpha_0 \) angle we can control the direction of flight.

4. Conclusions

Despite of the 2D simplification of the flapping motion the flows around of the profile are strongly non-linear and complicate. All phenomena depend strongly on the dynamic of vorticity. We noticed that disordered vorticity field around the flapping profile led to the disordered, unpredictable aerodynamic forces. In such case the stable flight is not possible.

References


Collapse vortices and filamentary structures

Henryk Kudela
Wrocław University of Technology, Faculty of Mechanical Engineering
Wybrzeże Wyspianskiego 27, 50-370 Wrocław, Poland
e-mail: henryk.kudela@pwr.edu.pl

Abstract

A system of a finite number of point vortices under suitable conditions related to the initial positions and circulations of the vortices and an assumption that the motions of vortices are in self-similar can collapse to the point with a finite time. It is described how to find numerically the initial positions leading to collapse. An explicit solution for a collapsing trajectory is derived. The numerical evidence is presented that collapsing vortices may organize themselves in a vortex sheet. In the distribution of vortices regular structures may be noticed. Examples are presented of the collapsing configurations with different number of vortices.

Keywords: point vortex, collapse, vortex sheet

1. Introduction

The study of dynamics of point vortices in a plane for an ideal incompressible fluid, has become an active field of research in the recent years. Collapse of vortices belongs to one of the most interesting problems related to the dynamics of vortices. Due to the fact that the distance between vortex changes during the motion, bringing the system to a different length scale collapse can be considered an elementary act in 2D turbulence kinetics. Equations of motion of the system of n-point vortices on the plane with distinct positions $z = (z_1, z_2, \ldots, z_n) \in \mathbb{C}^n$, $z_k = x_k + iy_k$, and circulations $\Gamma_1, \Gamma_2, \ldots, \Gamma_n$, each $\Gamma_j \in \mathbb{R} \setminus \{0\}$ are

\[
\frac{dz_k(t)}{dt} = v_k(z(t)) = \frac{i}{2\pi} \sum_{l \neq k} \Gamma_l \frac{z_k - z_l}{|z_k - z_l|^2}
\]

(1)

It is well-known that the systems (1) pose several invariants \[1, 4, 6\], that for a self-similar motion and some $\lambda(t)$ during the self-similar motion will connect to the point with a finite time. It is described how to find numerically the initial positions leading to collapse. An explicit solution for a collapsing trajectory is derived. The numerical evidence is presented that collapsing vortices may organize themselves in a vortex sheet. In the distribution of vortices regular structures may be noticed. Examples are presented of the collapsing configurations with different number of vortices.

Keywords: point vortex, collapse, vortex sheet

2. Self-similar motions and collapse of n vortices in the plane

The following definition was assumed of self-similar collapse motion of the n-vortices:

Definition. The system of the n-vortices is in self similar collapse motion if there exist complex function $\lambda(t) \in \mathbb{C}$, $\lambda(t) = \lambda_0(t) + i\lambda_1(t)$ that $Re(\lambda) = \lambda_0(t) < 0$ and $\lambda_j(0) \neq 0$, such that for all $k$ we have

\[
\frac{dz_k}{dt} = v_k = \lambda(t)z_k, \quad k = 1, 2, \ldots, n
\]

(3)

From (3) we have

\[
v_k(t) = v_j(t) = \lambda(t)(z_k(t) - z_j(t)), \quad k = 1, 2, \ldots, n
\]

(4)

In [4] it was proved that only $n - 3$ equations (4) are independent. It is possible to find the solution of the equation (3) in the form (1). Introducing new variables $(r(t), \varphi(t))$ and assuming that $z_k = z_k(0)r(t)e^{i\varphi(t)}$, $r(0) = 1$, $\varphi(0) = 0$, the solution takes the form [2, 3]

\[
z_k(t) = \sqrt{2\lambda_0(0)t + 1}e^{i\frac{\lambda_1(0)}{2\lambda_0(0)} \ln(2\lambda_0(0)t + 1)}} z_k(0)
\]

(5)

Solution (5) represents the logarithmic spiral. The critical (collision) time $t \rightarrow T_c$ is

\[
T_c = -\frac{1}{2\lambda_0(0)}
\]

(6)

If $\lambda_0(t) > 0$ the vortices system expand. In order to change the direction of the motion the sign of the circulations of the all vortices is changed to the opposite one [6]. If the real part of $\lambda(t)$ equals to zero, $\lambda_0 = 0$, the vortices are in relative equilibrium and the systems rotate as solid bodies, the collapse time is infinite, $T_c = \infty$.

Putting the solution $z_k = z_k(0)r(t)e^{i\varphi(t)}$ to the Hamiltonian $H$ in (2) it holds

\[
H(t) = H(0) - \frac{1}{2\pi} \ln |r(t)| \sum_{k, k \neq j} \Gamma_j \Gamma_k
\]

(7)

The Hamiltonian $H$ int(2) during the self-similar motion will conserve, when the virial $V = 0$. 

Due to the invariants of motion it seems that collapse of vortices contradicts the intuitive characteristic that the distance $r_{ij}$ between any two vortices cannot be much less than the smallest distance between any pair of vortices initially. But as it was explored for $n = 3$ in [1, 7], that for a self-similar motion and some additional conditions for circulations of the three vortices can collide in the center of vorticity in finite time. It will be demonstrated that such a collapse is possible for any $n \geq 3$. It will be given numerical evidence that the presence of the one or two bigger vortices than rest vortices in the collapsing set organized the point verities in filaments interpreted as vortex sheets [5]. That filamentary structures are fundamental in 2D turbulence.
The necessary conditions for collapsing of the vortex systems are \( V = 0 \) and because \( r_{ij} \) should go to 0 also \( L = 0 \). Without losing the generality it may be assumed that \( M_x = 0 \) and \( M_y = 0 \). Thus the vortex system will collapsed to the beginning of the coordinate system. It can be proved that when \( V = 0, M = 0 \) then also \( S = 0 \) and \( L = 0 \). From the similarity of the motion (3), is taken the following algebraic system of equations[4, 3]

\[
v_{k}z_k = v_{k}z_1 \quad (k = 1, \ldots, n - 3).
\]  

(8)

The collapsing positions of vortices is given by the \( V = 0, M_x = 0, \) and \( M_y = 0 \) and common zeros of \( f_1 = S \) and \( f_j = v_{j+2}z_{j+2} - v_{j+2}z_{j+1} = \cdots = n - 2 \). To complete the systems to 2n equations, it was assumed that one of the vortex in the system e.g \( z_n \) has a fixed position, it was included in the system of equations as an identity \( \sum \Gamma_j z_j^*v_j = 0 \) [3]. The nonlinear algebraic system of equation was solved by the Newton method [2, 3].

3. Examples of numerical results

In Figure 1 the collapsing system of 50 vortices was shown. The system is composed with 25 point vortex with circulations equal to 1, \( \Gamma_{26}^{1} \) to \( \Gamma_{50}^{1} \). and 25 vortices that have circulations equal to \(-\frac{4}{3}\), \( \Gamma_{26}^{3} \) to \( \Gamma_{50}^{3} \). Due to fact that the circulations in absolute values are not far from each other the distribution of collapsing vortices are spread around the center of vortices.

![Figure 1: Collapse of 50 vortices; \( \Gamma_{26}^{1} = 1 \) and \( \Gamma_{50}^{1} = -\frac{4}{3} \), \( T_{cryt} = 34.4079 \). In order to keep the readability of the graph it was shown only 8 trajectories: the four trajectories for the positive vortices and the four trajectories for the negative one. Bigger (blue) points mark the negative vortices](image1)

When system of collapsing vortices is introduced one, two or three vortices that have much larger circulations than the others, the vortices take on the filamentary structures. Numerical results are presented in Fig. 2, 3 and 4. The filamentary structures are visible in picture of the 2D turbulent flows.

![Figure 2: Collapse of 50 vortices; \( \Gamma_{26}^{1} = 1 \) and \( \Gamma_{50}^{1} = -\frac{4}{3} \), \( T_{cryt} = 34.4079 \). It was shown only 3 trajectories](image2)

3.1. Examples of numerical results

In Figure 1 the collapsing system of 50 vortices was shown. The system is composed with 25 point vortex with circulations equal to 1, \( \Gamma_{26}^{1} = 1 \) and \( \Gamma_{50}^{1} = -\frac{4}{3} \). Due to fact that the circulations in absolute values are not far from each other the distribution of collapsing vortices are spread around the center of vortices.

![Figure 3: Collapse of 72 vortices; \( \Gamma_{26}^{1} = 1 \) and \( \Gamma_{72}^{1} \approx \Gamma_{72}^{1} = -\frac{4}{3} \), \( T_{cryt} = 34.4079 \). In order to keep the readability of the graph it was shown only 8 trajectories: the four trajectories for the positive vortices and the four trajectories for the negative one. Bigger (blue) points mark the negative vortices](image3)

![Figure 4: Collapse of 72 vortices; \( \Gamma_{26}^{1} = 1 \) and \( \Gamma_{72}^{1} \approx \Gamma_{72}^{1} = -\frac{4}{3} \), \( T_{cryt} = 34.4079 \). In order to keep the readability of the graph it was shown only 8 trajectories: the four trajectories for the positive vortices and the four trajectories for the negative one. Bigger (blue) points mark the negative vortices](image4)

References

Vortex–in–cell method and parallel computations

Henryk Kudela¹, Andrzej Kosior²

¹,² Faculty of Mechanical and Power Engineering, Wroclaw University of Technology
Wybrzeze Wyspianskiego 27, 50-370 Wroclaw, Poland
e-mail: henryk.kudela@pwr.edu.pl ¹, andrzej.kosior@pwr.edu.pl ²

1. Introduction

The authors present numerical simulation of different vortex structures interactions with different initial conditions. The Vortex-In-Cell (VIC) method was used as a numerical algorithm. Due to problems with very long time of computations on the single processor parallel implementation of the VIC method on the multicore architecture of the graphics cards was created. Graphics Processing Units (GPUs) that were developed for video games can provide cheap and easily accessible hardware for scientific calculations.

The VIC method is well suited for parallel computation. The details of the authors’ implementation on a single GPU can be found in [2]. The main disadvantage of using graphics cards in computations is the limited amount of the RAM memory. To overcome this limitation a multiGPU implementation of the VIC method was created. Use of many GPUs in computations allowed for both: speedup and use of finer meshes in computations. The details on the implementation can be found in [3].

There are some common mechanisms in the reconnection process of two vortex tubes with nearly the same intensity. We noticed a typical sequence of physical events that was observed also by the others [1, 7]. First, the tubes tend to approach each other in antiparallel fashion advected by the mutual- and self-induction velocity. As the two vortex tubes get closer, the shape of the vortex core is deformed typically in to so-called head-tail structure. Then viscous cancellation of opposite signed vorticity in the interaction zone initiates vorticity reconnection. Adveeted by a complicated three-dimensional velocity field, the vorticity lines now experience a cross-linking, or bridging.

Equations of motion

Equations of incompressible and inviscid fluid motion have the following form:

\[
\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p
\]

(1)

\[
\nabla \cdot \mathbf{u} = 0
\]

(2)

where \( \mathbf{u} = (u, v, w) \) is velocity vector, \( \rho \) is fluid density, \( p \) is pressure. The equation (1) can be transformed into the Helmholtz vorticity transport equation:

\[
\frac{\partial \mathbf{\omega}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{\omega} = (\mathbf{\omega} \cdot \nabla) \mathbf{u}
\]

(3)

where \( \mathbf{\omega} = \nabla \times \mathbf{u} \). From incompressibility (2) stems the existence of vector potential \( \mathbf{A} \):

\[
\mathbf{u} = \nabla \times \mathbf{A}
\]

(4)

Assuming additionally that vector \( \mathbf{A} \) is incompressible (\( \nabla \cdot \mathbf{A} = 0 \)) its components can be obtained by solution of the Poisson equation

\[
\Delta A_i = -\omega_i, \quad i = 1, 2, 3
\]

(5)

Solving (5) one is able to calculate the velocity by formula (4). In vortex particle methods the viscous splitting algorithm is used. The solution is obtained in two steps: first, the inviscid - Euler equation is solved.

\[
\frac{\partial \mathbf{\omega}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{\omega} = (\mathbf{\omega} \cdot \nabla) \mathbf{u}
\]

(6)

In the second step the viscosity effect is simulated solving the diffusion equation.

\[
\frac{\partial \mathbf{\omega}}{\partial t} = \nu \Delta \mathbf{\omega}
\]

(7)

For the solution of the diffusive equation any suitable method like the Particle Strength Exchange (PSE) method or the Finite Difference method may be used.

Implementation on GPU

Graphics Processing Units that were developed for video games provide cheap and easily accessible hardware for scientific calculations.

An implementation of the Vortex Particle Method on GPU was done. Execution time for the application running on the GPU is 46 times shorter than the one using CPU. This is a very significant speed up. More details about this implementation can be found in [2].
2. Numerical examples

2.1. Vortex tubes reconnection case

In viscous fluid a test with reconnection of two vortex tubes was carried out. Test case was the same as one used in [7]. The vorticity field is assumed to be periodic with a period $2\pi$ in all the three orthogonal directions. We consider the motion of vortex rings in a cyclic cube of side $2\pi$. The computational grid had $256^3$ nodes. The test was with straight offset tubes of Gaussian cross section perpendicular to each other. Each vortex tube had the form

$$\omega(r) = \omega_0 \exp\left(-\frac{r^2}{l^2}\right)$$

(8)

where $r$ is the distance from the core centerline, $\omega_0$ is the maximum vorticity at the core center and $l$ is the $e^{-1}$-fold radius of the core. In this case $l = 3^{-1/3}$ and $\omega_0 = 20$.

Figure 1 includes results obtained in the current work. The evolution of the vortex rings is conveniently represented by iso-surfaces of vorticity norm $|\omega|$. They are plotted in the Fig. 1 at several representative stages of evolution. The level of the isosurface plotted is $|\omega| = 12$. The agreement with [7] is very good.

Figure 1: Three-dimensional Vortex Tube Reconnection

Figure 2: Stirring of passive markers during reconnection

2.2. Head-on collision of two vortex rings

The test shows the head-on collision of the two vortex rings. The experimental results were published in [5] and on the web side of Lim [4].

In this case the parameters were as follows $\omega_0 = 20$, $a = 0.5$, $R = 1.0$, $\nu = 0.01$, $Re = 1000$.

It is known that vorticity is not carried by the fluid particles in the viscous flow and the vorticity is not perfect means for visualization of the flow. Due to this, in order to better compare our results to the real flow visualization we used the passive markers carried by the fluid. The rings we merged in boxes of passive markers (colored with the initial position: red - lower ring, blue - upper ring). The size of the cuboids was equal to the diameter of the vortex rings and their thickness. Inside of the each box we put a grid of $100 \times 100 \times 50$ nodes and at each of the node a passive marker was put. The visualisation with passive markers is presented in Fig. 3. The structure that can be seen in the frame $t = 5$ resembles well the experimental pictures of Lim [4].

Figure 3: Head-on collision of two vortex rings visualized with passive markers

References

The meshless procedure for the stream function-vorticity formulation of the Navier-Stokes equations

Magdalena Mierzwiczak*

Institute of Applied Mechanics, Poznan University of Technology
pl. Marii Skłodowskiej-Curie, 60-695 Poznan, Poland
e-mail: magdalena.mierzwiczak@put.poznan.pl

Abstract

The meshless procedure is developed for the solution of the streamfunction-vorticity formulation of the Navier-Stokes equations. The fundamental solutions take into account the coupling between the differential equations through the vorticity term in the streamfunction equation. The non-linear terms are considered as inhomogeneities and treated by simple direct iteration. The Method of Fundamental Solutions (MFS) and the Method of Particular Solutions (MPS) are used to find the approximate solutions for the vorticity and the streamfunction at each iteration step. The flow inside a square cavity is used as a numerical example.

Key words: Streamfunction-Vorticity formulation, Navier-Stokes equations, Method of Fundamental Solutions

1. Introduction

The solution of the streamfunction-vorticity formulation of the Navier-Stokes equations via the Boundary Element Method (BEM) was performed in paper [5]. The authors developed the operator-splitting method to solve the primitive variables formulation of the Navier-Stokes equations. In paper [6] Liao and Zhu confirm that the high-order streamfunction-vorticity BEM is possible with the homotopy method together with Taylor series. In Refs. [1,2] the MPS procedure based on the homotopy method is proposed to solve the approximate solutions of boundary value problem (1-4) can be express as

\[ \alpha(x,y) = \sum_{j=1}^{N} \alpha_j G_\alpha(r) + \sum_{j=1}^{M} \beta_j \Phi(r) + \sum_{j=1}^{M} \gamma_j Q(x,y), \] (5)

\[ \psi(x,y) = \sum_{j=1}^{N} \alpha_j G_\psi(r) + \sum_{j=1}^{M} \alpha_j G_\psi(r) + \sum_{j=1}^{M} \beta_j \Phi(r) + \sum_{j=1}^{M} \gamma_j Q(x,y), \] (6)

where Re is the Reynolds number. For the MFS and the MPS, the approximate solutions of boundary value problem (1-4) can be express as

\[ \alpha(x,y) = \sum_{j=1}^{N} \alpha_j G_\alpha(r) + \sum_{j=1}^{M} \beta_j \Phi(r) + \sum_{j=1}^{M} \gamma_j Q(x,y), \] (5)

\[ \psi(x,y) = \sum_{j=1}^{N} \alpha_j G_\psi(r) + \sum_{j=1}^{M} \alpha_j G_\psi(r) + \sum_{j=1}^{M} \beta_j \Phi(r) + \sum_{j=1}^{M} \gamma_j Q(x,y), \] (6)

where \( G_\alpha(r) = \ln(r) \), \( G_\psi(r) = -\frac{r^2}{4} (\ln(r) - 1) \) are the fundamental solutions, \( \Phi(r) = \frac{1}{9} (4(c^2 + r^2) - r^2 - \frac{c^3}{3} \ln(c + \sqrt{c^2 + r^2}) \) is a particular solution for a Multiquadric function \( \phi(r) = \sqrt{r^2 + c^2} \) (c - shape parameter), that \( \nabla^2 \Phi = \phi \), and \( Q(x,y) \) are a particular solutions for monomials \( P(x,y) \), given in Table 1, that \( \nabla^2 Q = Q \).

Table 1: Monomials and their particular solution for Laplace’s operator

<table>
<thead>
<tr>
<th>( i )</th>
<th>( P_i )</th>
<th>( Q_{i,1} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>( x )</td>
<td>( (x^2 + y^2)^2/4 )</td>
</tr>
<tr>
<td>2</td>
<td>( x^2 )</td>
<td>( (x^2 + y^2)^2/8 )</td>
</tr>
<tr>
<td>3</td>
<td>( y )</td>
<td>( x(x^2 + y^2) )</td>
</tr>
<tr>
<td>4</td>
<td>( xy )</td>
<td>( x^2y(x^2 + y^2)/12 )</td>
</tr>
<tr>
<td>5</td>
<td>( x^2 )</td>
<td>( (x^2 + y^2)^2 \cdot y^2/14/14 )</td>
</tr>
<tr>
<td>6</td>
<td>( y^2 )</td>
<td>( (y^2 + x^2)^2 \cdot x^2/14/14 )</td>
</tr>
</tbody>
</table>

The unknown coefficients \( \{ \beta_{i,1}^{P}, \beta_{i,2}^{P}, \gamma_{i,1}^{P}, \gamma_{i,2}^{P} \} \) we obtain by interpolating then inhomogenous terms

\[ \sum_{j=1}^{M} \beta_j^{P} \phi(r_j) + \sum_{j=1}^{L} \gamma_j^{P} P_j(x_j, y_j) = f(x_j, y_j), j = 1, ..., M, \] (7)

\[ \sum_{j=1}^{M} \beta_j^{P} \phi(r_j) + \sum_{j=1}^{L} \gamma_j^{P} P_j(x_j, y_j) = 0, i = 1, ..., L, \] (8)

*The work has been supported by 2012/07/B/ST8/03449 Grant.
where \( r_j = \sqrt{(x_j - x)^2 + (y_j - y)^2} \), \( \{x_j, y_j\}_{j=1}^M \) are interpolation points distributed uniformly in computational domain \( \Omega \) (see Fig.1),

\[
g(x_j, y_j) = -\sum_{i=1}^{M} \beta_j^i \Phi(x_j, y_j) - \sum_{i=1}^{N^c} \gamma_i^j \Psi(x_j, y_j).
\]

Figure 1: Geometry with ○ collocation, ● source and □ interpolation points, and boundary conditions for flow inside a square cavity

Figure 2: Contour of vorticity and streamfunction for Re=\{0, 50, 100\}

The unknown coefficients \( \{\alpha_i^N, \beta_i^N\}_{i=1}^{N} \) we obtain by collocating the boundary conditions (3) and (4)

\[
\sum_{i=1}^{N} \alpha_j^i \Phi(x_j, y_j) + \sum_{i=1}^{N} \beta_j^i \Psi(x_j, y_j) = \psi_j - \sum_{i=1}^{L} \Phi(x_j, y_j), \quad (9)
\]

\[
\sum_{i=1}^{N} \alpha_j^i \frac{\partial \Phi}{\partial n}(x_j, y_j) + \sum_{i=1}^{N} \beta_j^i \frac{\partial \Psi}{\partial n}(x_j, y_j) = \frac{\partial \psi_j}{\partial n} - \sum_{i=1}^{L} \frac{\partial \Phi}{\partial n}(x_j, y_j), \quad (10)
\]

where \( r_s = \sqrt{(x_s - x)^2 + (y_s - y)^2} \), \( \{x_s, y_s\}_{s=1}^{N^s} \) are source points located outside computational domain \( \Omega \) on the fictitious boundary and \( \{x_j, y_j\}_{j=1}^{N^c} \) are collocated points distributed on the boundary \( \Gamma \) (see Fig. 1).

3. Numerical applications

In order to examine the presented scheme, consider the cavity flow problem. The geometry and the corresponding boundary conditions are shown in Fig.1. The results corresponding to the Reynolds number of 0, 50 and 100 are shown in Fig. 2. The result was obtain for \( N = 104, N^c = 204, M = 625, L = 6, s = 0.1, c = 0.01 \). For Re = 0 we not need the iteration. For Re = 50 and Re = 100 we obtained the results by simple direct iteration.

4. Conclusions

The meshless procedure is applied to solve the streamfunction-vorticity formulation of the Navier-Stokes equations. The fundamental solutions take into account the coupling between the differential equations through the vorticity term in the streamfunction equation. The formulation avoid calculation of the vorticity values on the boundaries. The non-linear terms are considered as inhomogeneities at each iteration step and interpolated by the radial basis functions.

References


Discrete Element Method in the influence study of faults of concrete specimens on uniaxial compression test

Tomasz Nowicki
Department of Structural Mechanics, Faculty of Civil Engineering and Architecture, Lublin University of Technology
ul. Nadbystrzycka 40, 20-618 Lublin, Poland
e-mail: t.nowicki@pollub.pl

Abstract

Numerical models based on three dimensional Discrete Element Method were used to study the behaviour of concrete under standard uniaxial compression tests. The practical side of the research focuses on faults and defects that can disturb the tests and affect final results. Numerical part of the work is a study on usefulness of Discrete Element Method for modelling concrete material for engineering purposes.

Keywords: Discrete element method; Concrete; Uniaxial compression test

1. Introduction

It is known for over two thousand years, concrete is constantly under interest of researchers. This is due to the fact that the name 'concrete' covers a very wide spectrum of physical and chemical phenomena which packaged under the name are ubiquitous in a human environment as one of the most common building material. This paper presents results of ongoing study on Discrete Element Method (DEM) suitability to model structural plain and reinforced concrete. The method has been chosen as an alternative to Finite Element Method, which has disappointed in the area due to the difficulties in encompassing effects of fracture. The presented research focuses on a uniaxial compression test. The test is one of the most common laboratory experiments carried out at universities and in construction companies.

2. Research program

The research in its broader picture is intended to be twofold. Firstly DEM is investigated as a numerical modelling tool for engineering purpose. Secondly a technical problem of defective laboratory concrete specimens was undertaken. Typical concrete specimens are presented in Fig. 1. If the specimen comes from a real structure it is usually of cylindrical shape (Fig. 1b) because the shape is easier to obtain. Such samples are always at risk of existing hidden faults such as cracks, voids or big fragments of aggregate. The defects obviously influence the laboratory tests changing their results and not discovered in time may lead to wrong decisions.

The ongoing research program covers numerical modelling of different specimens with or without faults. Laboratory experiments are planned as well.

2.1. Sample results

The presented simulation concerns a cubic sample (Fig. 2) with edge length a=h=0.1m (according to Fig. 1). This sample was examined in two variants. In the first case (named A) the cube was a solid block, which represented a specimen without any defects. In the second case there was a spherical void in the interior of the cube.

Figure 1: Typical specimen shapes used for uniaxial compression tests of concrete: a) cuboid, b) cylinder: 1,2 outline, 3 spherical hidden void as a specimen fault, 4 pre-existing cracks

Figure 2: Cross-section of a cubic sample of size a=0.1m used in the presented simulation: 1 –material bulk, 2 –spherical vacuum, 3 –immobile plane, 4 –plane moving at the speed of u

In computer simulation the specimen was a DEM numerical model (Fig. 3 and 4) consisting of 1122 or 1004 spheres according to the variant. The porosity level was about 40%. Maximum and minimum radius of the sphere set was 0.012m and 0.003m, respectively.
The specimens were generated using a geometrical generator Random Box Packer available in ESys–Particle software package [5]. Computer simulations were carried out using Yade DEM framework [6].

During computer simulation the specimen was virtually placed between two non–deformable plates. The bottom one was stationary and acted on the specimen on the basis of action–reaction law only. The top plate was the active one. It moved downwards at constant velocity of \( u=0.1\text{m/s} \). Although this rate was higher than that used in laboratory experiments, it was still sufficient small to maintain quasi–static conditions i.e. the velocity was significantly lower than compressional wave speed in the simulated sample. Moreover the target velocity was achieved within the first 2000 simulation steps with constant acceleration starting form \( u=0.0\text{m/s} \) at the beginning of the simulation.

The concrete material was modelled with CMP (Concrete Particle Model) material model available in Yade package. The model replaces material continuity with pre–established cohesive contacts between particles. The model features. damage in tension, plasticity in shear and compression. The plates were modelled as elastic material with friction.

Figures 3 and 4 present snapshots of computer simulations. The first one shows the specimen at the early state of the test, the second one in the final stage of experiment. The change in colour, i.e. transition from blue to purple, visualises effort of materials. The diagram in Fig. 5 is a strain–stress curve obtain in the simulation.

The results presented here show that a material model that was applied in the simulations, despite its simplicity, provided a valuable data. The stress and strain values concern the cubic specimen as a whole. The stress is calculated dividing the resultant force acting on a wall (top or bottom) of the specimen by the area of the wall. The strain is a ratio of a change in distance of the top and bottom walls to their initial position.

Firstly the elastic phase is clearly visible, which allows to find Young’s modulus of the whole specimen easily. The plastic phase obtained in the simulation is shorter then in laboratory tests. However it is still possible to estimate the strength of the material that is 6 MPa here. The diagram shows also that relatively big vacuum in the specimen may not be discovered in standard uniaxial compression test. The phenomenon is an object of ongoing study started with this research.

3. Final remarks

A Discrete Element Method is undoubtedly well suited for modelling the behaviour of concrete material in both simple quasi–static and dynamic load. The ability was documented in many papers e.g. [2,3,4], but the results are always product of the discrete material model employed in numerical simulation. Suitability of the models for engineering applications and their verifications is going to be the main matter presented on PCM-CMM conference.

References

[5] https://launchpad.net/esys-particle
Application of the Okubo-Weiss parameter to dynamical resolution adjustment in the Smoothed Particle Hydrodynamics approach

Michał Olejnik1∗, Kamil Szewc2∗, Jacek Pozorski3∗

1,2,3 Institute of Fluid-Flow Machinery, Polish Academy of Sciences
Fiszera 14, 80-231 Gdańsk, Poland
e-mail: michal.olejnik@imp.gda.pl

1 Conjoint Doctoral School of Gdańsk University of Technology and IFFM PAS
Narutowicza 11/12, 80-233 Gdańsk, Poland

Abstract

In the work a novel method is proposed of dynamical resolution adjustment by means of modification of the smoothing length parameter in the SPH approach. Although a local mesh refinement poses no vital problems in grid based methods, it is not so simple for particle methods as SPH. In order to detect regions where changes of resolution are necessary, the application of the Okubo-Weiss parameter is proposed. Our approach is validated using a two-dimensional Taylor-Green vortex.

Keywords: gridless methods, Smoothed Particle Hydrodynamics, Okubo-Weiss parameter, adaptivity, smoothing length refinement

1. Introduction

The Smoothed Particle Hydrodynamics (SPH) is a gridless, particle based method of fluid flow modelling. Firstly, it was developed in order to model astrophysical phenomena. In recent years it has been successfully applied in simulations of fluid and solid mechanics. The main idea behind SPH is to discretize the domain into a set of particles and to interpolate field values at each node (particle) using a weighting kernel function \( W \). The range of this kernel is usually denoted by \( h \) and called the smoothing length. The \( \Delta r \) and \( h \), where \( \Delta r \) is the mean distance between particles, are the main parameters yielding by the accuracy of SPH simulations. The ratio \( h/\Delta r \) acts strongly on the quality of results. Since SPH is a Lagrangian approach, its local refinement is harder than in the grid based methods. Algorithms exist for the change of a local number and distribution of particles according to density or velocity gradient, see [2, 3]. In the work we present a novel method of dynamical adjustment of \( h \), while \( N \) remains constant, to imply changes of \( h/\Delta r \). In this approach a criterion determining the value of \( h \) is the Okubo-Weiss parameter \( \sigma \) — widely used in studying sea eddies and currents. It divides flow into regions dominated by vorticity or strain.

2. SPH formulation and fluid flow equations

Let us assume that for a field \( A \) the relationship holds

\[
A(r) = \int_{\Omega} A(r') \delta(r - r') \, dr',
\]

where \( r \) denotes position in domain \( \Omega \) and \( \delta(r) \) is Dirac delta function. In order to obtain SPH interpolation we replace \( \delta(r) \) with a normalized, symmetric kernel function \( W(r, h) \) which converges to \( \delta(r) \) with \( h \to 0 \) and vanishes for \( |r| > 2h \) (depending on the kernel choice). Next, the domain is divided into a set of volumes \( \Omega_b = m_b/\rho_0 \), where \( m_b \) is the mass, \( \rho_0 \) is the density. The summation interpolation lying at the very basis of SPH approach is given as

\[
\langle A \rangle(r) := \sum_b A(r_b) \Omega_b W(r - r_b, h).
\]

Approximating the derivative of any variable is simply done replacing the kernel function with its derivative in Eqn (2). For a viscous, incompressible fluid flow a set of partial differential equations is used consisting of: the Navier-Stokes equation

\[
\frac{du}{dt} = -\frac{1}{\rho} \nabla p + \frac{\mu}{\rho} \Delta u + f,
\]

where \( u \) denotes the velocity, \( t \) the time, \( p \) the pressure, \( \mu \) the dynamic viscosity and \( f \) the external acceleration; the continuity equation

\[
\frac{d\rho}{dt} = -\rho \cdot u \xrightarrow{\text{approx.}} \nabla \cdot u = 0,
\]

and the advection equation

\[
\frac{dr}{dt} = u.
\]

In order to ensure that the incompressibility condition is satisfied the most common approach is applied, called Weakly Compressible SPH (WCSPH). In this method the set of governing equations is closed with the equation of state in the form

\[
p = \frac{c^2 \rho_0}{\gamma} \left( \frac{\rho}{\rho_0} \right)^{\gamma - 1},
\]

where \( c \) is the speed of sound, \( \gamma \) is a constant and \( \rho_0 \) is the reference density. To keep density fluctuations at the level of 1% we set \( c \) to speed 10 times higher than maximum speed in the flow and \( \gamma \) to 7. Using SPH interpolation, the equations governing the flow can be solved for each particle to obtain solution for the whole domain. A detailed description of SPH method can be found in [4].

∗This research was supported by the National Science Centre (NCN, Cracow, Poland) through the research project 2011/03/B/ST8/05677 and by the EU FP7 Nugenia-Plus project (grant agreement No. 604965).
3. The Okubo-Weiss parameter

3.1. Definition and meaning

The Okubo-Weiss parameter is commonly used in the research of sea and ocean flows. We define it as in [1] by
\[
\sigma := s_n^2 + s_s^2 - \omega^2,
\]
where \(s_n\) and \(s_s\) are normal and shear components of the strain rate tensor, while \(\omega\) is vorticity. For a two-dimensional flow with the velocity field \(\mathbf{u} = [u, v]\) they are given as
\[
s_n = \frac{\partial u}{\partial x} \frac{\partial v}{\partial y}, \quad s_s = \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} - \frac{\partial u}{\partial x} \frac{\partial v}{\partial y}, \quad \omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}.
\]
The value \(\sigma\) characterizes nature of flow in a given part of considered domain. If \(\sigma < 0\), then vorticity overcomes strain, if \(\sigma > 0\) it is the other way round.

3.2. Idea behind application in SPH

The idea to use the Okubo-Weiss parameter in SPH arose during the analysis of a two-dimensional Taylor-Green vortex (TGV). In this case a periodic array of decaying vortices in \(x - y\) plane occurs. The velocity field at \(t = 0\) (dimensionless with the time scale \(L/u_{\text{max}}\), where \(u_{\text{max}}\) is maximum velocity) is given by
\[
\begin{align*}
    u(x, y) &= -u_{\text{max}} \cos(2\pi x/L) \sin(2\pi y/L), \\
    v(x, y) &= u_{\text{max}} \sin(2\pi x/L) \cos(2\pi y/L).
\end{align*}
\]
We considered \(0 \leq x, y \leq L\) with fully periodic boundary conditions and the Reynolds number \(Re = 100\). Starting from the uniform distribution of particles in the domain we analysed the flow (particle positions). The greatest deviation from the analytical trajectories was observed for particles initially placed in the regions dominated by strain. The error analysis tended to show smaller discrepancy in the neighbourhood of vortices. This was correlated with the value of \(\sigma\), whose distribution is showed in Fig. 1.

![Figure 1: The map of the Okubo-Weiss parameter in TGV at \(t = 0.1\)](image)

Furthermore, increasing \(N\) (with a constant ratio of \(h/\Delta r\)) has not lead to better results. On the other hand a higher value of \(h/\Delta r\) yields evident improvement. Due to these observations we decided to set \(h\) for each particle separately, depending on \(\sigma\) value in the following manner: minimum and maximum \(h\) \((h_{\text{min}}\) and \(h_{\text{max}}\)) correspond to minimum and maximum values of the Okubo-Weiss parameter in the domain \((\sigma_{\text{min}}\) and \(\sigma_{\text{max}}\)), while transition between \(h_{\text{min}}\) and \(h_{\text{max}}\) is described by a continuous function. This method of \(h\)—adjustment is easy to implement in an SPH formulation and fits with its Lagrangian nature.

4. Results

The validation case of the algorithm was the TGV, already described in Section 3.2. In order to test the accuracy of the approach three SPH simulations were performed for \(N = 4096\) particles, with \(h_{\text{max}} = 1/16\), \(h_{\text{min}} = 1/32\) and with \(h\) linearly dependent on the value of \(\sigma\):
\[
h(\sigma) = \frac{\sigma - \sigma_{\text{min}}}{\sigma_{\text{max}} - \sigma_{\text{min}}} (h_{\text{max}} - h_{\text{min}}) + h_{\text{min}}.
\]

Figure 2 compares evolution of the total kinetic energy in the whole domain obtained through the SPH simulations and the analytical prediction. The worst results were obtained for \(h = 1/32\), while increasing its value to \(1/16\) gave a significant improvement. The latter ones, however, are barely distinguishable from the results of simulation with a variable \(h\) according to Eqn (10). This proves that the increasing \(h\) for regions generating error locally improves the accuracy of calculations.

![Figure 2: The evolution of the total kinetic energy in TGV; solid line denotes analytical prediction](image)

5. Conclusions

The proposed method of adjusting the smoothing length \(h\) according to distribution of the Okubo-Weiss parameter is a promising to increase the quality of SPH simulations. The obtained results for the TGV test case are accurate, staying in agreement with the analytical predictions. The considered case, however, is relatively simple. For this reason validation for more complicated cases is necessary. Currently the algorithm is tested for a more complex flow, like the lid-driven cavity and simple multiphase flows. Initial results are promising showing potential for further development. A question how the choice of function \(h(\sigma)\) influences performance and computational cost deserves a deeper insight too.

References


Modelling of transient heat transport in a two-layered crystalline solid films using the interval lattice Boltzmann method

Alicja Piasecka - Belkhayat¹, Anna Korczak²
Institute of Computational Mechanics and Engineering, Silesian University of Technology, Konarskiego 18A, 44-100 Gliwice, Poland
e-mail: alicja.piasecka@polsl.pl ¹, anna.korczak@polsl.pl ²

Abstract

In the paper numerical modelling of heat transfer in one-dimensional crystalline solid films is considered. A generalized two-layer problem is described by Boltzmann transport equations transformed in the phonon energy density equations supplemented by the adequate boundary-initial conditions. Such approach in which the parameters appearing in the problem analyzed are constant values is widely used, but in the paper the interval values of relaxation time and boundary condition for silicon and diamond are taken into account. The problem formulated was solved by means of the interval lattice Boltzmann method using the rules of directed interval arithmetic. In the final part of the paper the results of numerical computations are presented.

Keywords: Boltzmann transport equation, interval lattice Boltzmann method, directed interval arithmetic

1. Introduction

In dielectric materials and semiconductors the heat transport is mainly performed by quanta of lattice vibrations called phonons. The phonons represent the conduction of heat and electricity through solids. In non-metals phonons as heat carriers always “move” from the higher temperature part to the lower temperature part and during this motion phonons carry energy. This kind of phenomena can be described by the Boltzmann transport equation (BTE). Such approach in which the parameters appearing in the mathematical model are constant values is widely used [1,3]. Here, the interval values of relaxation times and boundary conditions for successive sub-domains are taken into account. The relaxation time is estimated experimentally and its actual value is still a subject of discussion [5]. In the paper heat transport proceeding in a two-layered thin film is considered [2,7]. In order to solve the problem the interval lattice Boltzmann method is applied using the rules of directed interval arithmetic [4,6]. In the final part of the paper examples of numerical computations are shown.

2. Boltzmann transport equation

The unsteady BTE in a phonon energy density formulation using the simplifying assumptions of the Debye model for one-dimensional two-layered analysis [1] can be written as

\[
\frac{\partial e_s}{\partial t} + v_s \nabla e_s = - \frac{e_s}{\tau_{rs}} + q_{vs}
\]  

(1)

where \( s = 1, 2 \) corresponds to the successive layers of a thin film (silicon, diamond), \( e_s \) is the phonon energy density, \( e^0_s \) is the equilibrium phonon energy density, \( v_s \) is the frequency-dependent phonon propagation velocity, \( \tau_{rs} \) is the frequency-dependent phonon relaxation time, \( t \) denotes time and \( q_{vs} \) is the internal heat generation rate related to an unit volume.

Using the Debye model the dependence between phonon energy and lattice temperature can be calculated from the following formula

\[
e_s(T) = \left( \frac{\eta_s k_s}{\Theta_{Ds}} \right) \int_0^{\infty} \exp\left(\frac{z^3}{3}\right) T z dz\n\]

(2)

where \( \Theta_{Ds} \) is the Debye temperature of the solid, \( k_s \) is the Boltzmann constant, \( T_s \) is the lattice temperature while \( \eta_s \) is the number density of oscillators [1].

The equations (1) should be supplemented by boundary and initial conditions.

3. Interval lattice Boltzmann equation

The lattice Boltzmann method (LBM) is a numerical technique for the simulation of heat transfer. The LBM solves a discretized set of the BTE known as the lattice Boltzmann equations. The phonon energy density is defined as the sum [1]

\[
e_s(x, t) = e_{s1}(x, t) + e_{s2}(x, t) = \sum_{d=1}^{2} e_{sd}(x, t)
\]

(3)

where \( e_{sd} \) is the phonon energy density in the positive \( x \) direction for \( s \)th layer while \( e_{s2} \) is the phonon energy density in the negative \( x \) direction, \( d \) means the lattice direction.

The interval Boltzmann transport equations for a one-dimensional problem take the form [6]

\[
\frac{\partial e_{sd}}{\partial t} + v_s \frac{\partial e_{sd}}{\partial x} = \frac{e_{sd}}{\tau_{rs}} - \frac{e^0_{sd}}{\tau_{rs}} + q_{vs}
\]

(4)

where \( v_s = \Delta x_s / \Delta t \) is the velocity component along the \( x \)-axis, \( \Delta x_s \) is the lattice distance from site to site, \( \Delta t = t^{t+1} - t^t \) is the time step needed for a phonon to travel from one lattice site to
the neighboring lattice site, \( \tau_{rs} = [\tau_{rs}^l, \tau_{rs}^r] \) is the interval relaxation time and \( e_{\tau_{rs}}^r = e_{\tau_{rs}}(x, t) / d \).

The set of equations (4) must be supplemented by the boundary-initial conditions [2,7]

\[
\begin{align*}
&x = 0: \quad \tau_{rs}(0, t) = \tau_{rs}(T_{rsi}) \\
&x = L: \quad \tau_{rs}(L, t) = \tau_{rs}(T_{rsi}) \\
&t = 0: \quad e_{\tau_{rs}}(x, 0) = e_{\tau_{rs}}(T_{rsi})
\end{align*}
\]

(5)

where \( T_{rsi} \) and \( T_{rsi} \) are the interval boundary temperatures, \( T_{rs} \) is the initial temperature. Between the successive sub-domains the continuity condition can be taken into account [7]

\[
\begin{align*}
&x = L/2: \quad \tau_{rs}(x, t) = \tau_{rs}(x, t)
\end{align*}
\]

(6)

The interval LBM algorithm has been used to solve the problem analyzed [4,6]. The approximate form of the equations (4) is of the following form

\[
\begin{align*}
\tau_{rs}^{l+1} = (1 - \Delta t / \tau_{rs}^l) \bigg( \tau_{rs}^l \bigg)^{\Delta t / \tau_{rs}^r} \bigg( \tau_{rs}^r \bigg)^{\Delta t / \tau_{rs}^r} + \Delta t q_{rs} \\
\tau_{rs}^{2+1} = (1 - \Delta t / \tau_{rs}^l) \bigg( \tau_{rs}^l \bigg)^{\Delta t / \tau_{rs}^r} \bigg( \tau_{rs}^r \bigg)^{\Delta t / \tau_{rs}^r} + \Delta t q_{rs}
\end{align*}
\]

(7)

After subsequent computations the interval lattice temperature is determined using the formula (see Eqn (2))

\[
T_{rs}^{l+1} = \frac{1}{4} \sum_{rs} \left( \sum_{rs} \frac{\Theta_{rs}^l}{\Theta_{rs}^l + \int_0^x \frac{z^2}{\exp(z) - 1} \, dz} \right)
\]

(8)

4. Results of computations

As a numerical example the heat transport in a silicon-diamond film of the dimension \( L=200 \) nm was analyzed. The following input data were introduced for a silicon-diamond film:

\[
\begin{align*}
&\tau_{rs}^l = [6.37, 6.69] \text{ ps} \quad \tau_{rs}^r = [20.38, 21.42] \text{ ps} \quad \Theta_{rs}^l = 640 \text{ K} \quad \Theta_{rs}^r = 2200 \text{ K} \\
&T_{rsi} = [780, 820] \text{ K} \quad T_{rsi} = [292.5, 307.5] \text{ K} \quad T_{rs} = 300 \text{ K} \\
&q_{rs} = 0 \quad \Delta x = 20 \text{ nm} \quad \Delta t = 5 \text{ ps}
\end{align*}
\]

4. Results of computations

As a numerical example the heat transport in a silicon-diamond film of the dimension \( L=200 \) nm was analyzed. The following input data were introduced for a silicon-diamond film:

\[
\begin{align*}
&\tau_{rs}^l = [6.37, 6.69] \text{ ps} \quad \tau_{rs}^r = [20.38, 21.42] \text{ ps} \quad \Theta_{rs}^l = 640 \text{ K} \quad \Theta_{rs}^r = 2200 \text{ K} \\
&T_{rsi} = [780, 820] \text{ K} \quad T_{rsi} = [292.5, 307.5] \text{ K} \quad T_{rs} = 300 \text{ K} \\
&q_{rs} = 0 \quad \Delta x = 20 \text{ nm} \quad \Delta t = 5 \text{ ps}
\end{align*}
\]

Figure 1 illustrates the interval temperature distribution in the domain considered for the chosen times. Figure 2 presents the courses of the temperature function at the internal nodes \( x_1 = 20 \) nm (1) and \( x_2 = 60 \) nm (2) for the silicon layer.

Figure 2: The interval heating curves at internal nodes

The generalization of LBM allows to find the numerical solution in the interval form and such an information may be important especially for the parameters which are estimated experimentally, for example the relaxation time.

The problem analyzed can be extended to multi-layered thin films.

References


Highly resolved LBM flow simulations in ceramic foams with experimental verification

Wojciech Regulski¹, Jacek Szumbarski², Łukasz Łaniewski-Wołłk³, Konrad Gumowski¹, Jakub Skibiński², Michał Wichrowski⁶

¹²³ Institute of Aerodynamics and Applied Mechanics, Warsaw University of Technology
Nowowiejska 24, 00-665 Warszawa, Poland
e-mail: wregulski@meil.pw.edu.pl

Financed by the Polish National Centre for Research and Development under the grant No. PCh-15-035/A/2015.

² Warsaw University of Technology
Woloska 141, 02-507 Warszawa, Poland

³ Faculty of Materials Engineering, Warsaw University of Technology
Wołoska 141, 02-507 Warszawa, Poland

⁶ Institute of Fundamental Technological Research, Polish Academy of Sciences
Pawińska 5, 02-106 Warszawa, Poland

Abstract

This work presents the results of numerical and experimental investigations of morphology (porosity and specific surface area) and pressured drop of a series of ceramic foam specimens. Morphology assessment is performed on post-processed computer-tomography (CT) images. The pressure drop of 10ppi (pore per inch), 20ppi and 30ppi foams with porosity in the range of 75% to 79% is examined using the D3Q19 multiple-relaxation-time lattice Boltzmann method (MRT-LBM). These results are compared against experimental data from measurements in the water channel. Comparison of numerical and experimental data exhibits very good agreement. Moreover, obtained results (foam parameters and pressure drops) are verified against other researchers’ data and correlations with varying outcomes.

Keywords: ceramic foams, pressure drop, Darcy-Forchheimer law, specific surface area, pore-scale simulation

1. Introduction

The specific properties of highly porous ceramic materials such as huge specific surface area, thermal and corrosion resistance and relatively low pressure drop make them well suited in a range of industrial applications such as filtration, reaction catalyst support or flow stabilisation. That results in a need for an accurate a priori determination of the pressure drop of these structures. The pressure drop in porous media is given by the Darcy-Forchheimer relation,

\[ \frac{\Delta p}{\Delta L} = \frac{\mu}{k_1} U + \frac{\rho}{k_2} U^2. \] (1)

Ideally, the viscous and inertial permeability coefficients \( k_1 \) and \( k_2 \) should be the function of the foams’ geometry [3]. In the case of this work, they are expressed as functions of porosity, \( \varepsilon \), and specific surface area, \( a_c \).

2. Measurement methods

The foam data was obtained from post-processed CT images. This procedure consisted from a few steps. First, foam specimens, each of 50×50×50mm in size, were scanned with proper resolution, then an appropriate filtering was applied. It was followed by the binarisation. Next, the spurious inner strut porosity was removed and the relevant foam data was calculated. The foam data is given in Table 1. It is important to emphasize that the digitized foam geometry has the voxel structure. The rendered image of the 10ppi foam specimen is shown in Fig. 1.

Once the foam geometries were obtained, porosity and specific surface area were found. The geometries in their voxelized form were used as input data to the LBM solver in order to determine the hydraulic resistance. Additionally, the original specimens were investigated in the water channel.

2.1. Measurement of specific surface area

Calculation of the structure surface area involves using dedicated algorithms. Two approaches were used. The first one was the Cauchy-Crofton formula that links the surface area of the object with the number of its cross-sections with the set of randomly generated lines [6], the second was based on established relation between local 2×2×2 voxel configurations and their area [5]. Both methods yield results that agree closely (less than 1% discrepancy).

2.2. Experimental apparatus

The foam specimens were investigated in the dedicated water channel with 50×50mm square cross-section. The flow was smoothed by a honeycomb straightener and developed over 400mm length before it reached the specimen. The flow rate control as well as pressure drop and velocity measurements were performed automatically. The velocities in the range of 0.007-0.2 m/s and pressure drops up to 6000 Pa were investigated.

2.3. The lattice Boltzmann method

The lattice Boltzmann method is a mesoscopic kinetic formulation of the flow dynamics. In the LBM macroscopic flow variables are reconstructed from particle populations \( f_i \) (\( i=0,1,\ldots,N-1 \)) that move on the Cartesian grid with fixed velocities \( c_i \). The LBM governing equation used in this work takes the form,

\[ f_i(t+\Delta t,x_i+c_i\Delta t) - f_i(t,x_i) = -M^{-1}S\left(f_i-f_i^{eq}\right), \] (2)

where the left hand side describes streaming of populations while the right hand side is the collision operator. The matrix \( M \) transforms populations to the hydrodynamic moment space and the diagonal matrix \( S \) contains relaxation rates that are connected to the transport coefficients. This formulation is known as the Multiple Relaxation Time (MRT) [1], it is used in three dimensions with 19 velocities (the D3Q19 MRT model).
Table 1: Parameters of ceramic foams based on reconstructed computer-tomography images. Each foam specimen consists of 672×672×800 voxels. Specific surface area obtained using Cauchy-Crofton formula [6].

<table>
<thead>
<tr>
<th>ppi number</th>
<th>scan-resolution [μm]</th>
<th>Physical dimensions [mm]</th>
<th>Porosity [-]</th>
<th>Specific surface area [m²/m³]</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>45.252</td>
<td>30.41×30.41×36.20</td>
<td>0.777</td>
<td>721.5</td>
</tr>
<tr>
<td>20</td>
<td>45.252</td>
<td>30.41×30.41×36.20</td>
<td>0.751</td>
<td>1075.0</td>
</tr>
<tr>
<td>30</td>
<td>25</td>
<td>16.8×16.8×20</td>
<td>0.787</td>
<td>1361.7</td>
</tr>
</tbody>
</table>

2.4. Solver and simulation setup

On contemporary computational hardware the LBM would not be efficient without proper parallelisation. In this case the optimized CUDA-C code for Graphical Processing Units (GPU) was used. The code exploits dedicated mechanisms such as memory coalescence. Additionally, the code was optimized by using symbolic operations and case-specific generation. A very good scalability up to 48 GPUs was observed, with typical simulation using half of those resources.

The foam specimens were confined in a channel with side walls and periodic inlet/outlet boundary conditions. Although the inlet/outlet geometries are non-overlapping, this effect turned out to be insignificant and simulations were carried out successfully with body force as driving mechanism. Pressure drop was measured along three perpendicular directions.

3. Simulation Results

The pressure drop results of 10ppi and 20ppi foam match the experimental results very well. The discrepancy is less than 10-15% depending on the flow direction. In contrast, the results for the 30ppi foam exhibit much bigger discrepancy. It is attributed to the heterogeneity of the foam structure and the fact that the size of the specimen was smaller than in the case of 10ppi and 20ppi foams and, thus, less representative. Additionally, the anisotropy of the foam structure is reported in all cases. The results for the 10ppi foam are shown in the Fig. 2.

3.1. Comparison to existing correlations

It is shown that the correlations proposed in [2] and [3], when used only with the specific surface area as the specific dimension, reduce to a single expression of the form,

\[ \frac{\Delta p}{M} = A_{1} \frac{\eta}{\rho U} + A_{2} \frac{\rho U}{2}, \]

(3)

where \( A_{1}=6.88 \) and \( A_{2}=0.36 \) in the case of [2] and \( A_{1}=6.25 \) and \( A_{2}=0.29 \) for [3]. Both correlations yield results that fall within the range spanned by our data with exception to [3] in the case of 30ppi foam that should be considered not representative.

4. Summary

The presented work demonstrates the capability of the LBM to resolve intricate flow patterns at pore-scale level and to predict pressure drop in porous media. The presented pressure drop data shows good agreement with our experimental results and existing correlations. The foam morphology data is well within the range of data reported in the literature [4].

References

Is the motion of a single SPH particle droplet/solid physically correct?

Kamil Szewc¹, Katarzyna Walczewska-Szewc², Michał Olejnik³

¹,²Institute of Fluid-Flow Machinery, Polish Academy of Sciences, ul. Fiszera 14, 80231 Gdańsk, Poland
e-mail: kszewc@imp.gda.pl¹
²In-Silico, Numerical Laboratory, ul. Jodowa 1B/5, 80680 Gdańsk, Poland

Abstract

In many situations, due to both: physical processes and numerical errors, the interface in multi-phase flows and the surface in free-surface flows can be fragmented. As a result, the single SPH particle solids/droplets (or even bubbles) can appear and move in the opposite phase or travel without any friction through the regions without particles (representing gaseous phase in free-surface flows). In the work we focus on the problem of physical correctness of such single particle solids/droplets motion. We try to answer the following questions: is the motion of a single particle droplet physically correct? What is such a droplet diameter? What is an influence of the SPH numerical parameters such as $h$, $h/\Delta r$ or the shape of the kernel on the motion of the single particle? Next, we propose two different remedies to the problems related to the interface fragmentation: the first to minimize the fragmentation process, the other to filter single SPH particles of one phase moving in the opposite phase (effects of fragmentation).

Keywords: SPH, mesh-free methods, multiphase flow, dispersed phase, interface fragmentation

1. Introduction

The Smoothed Particle Hydrodynamics method (SPH) is a mesh-free, particle-based approach for a fluid-flow modelling. In the early stage it was developed to simulate some astrophysical phenomena at the hydrodynamic level. The main idea behind the SPH method is to introduce kernel interpolants for flow quantities in order to represent fluid dynamics by a set of particle evolution equations. Due to its Lagrangian nature, for multiphase flows, there is no necessity to handle (reconstruct or track) the interface shape as in the grid-based methods. Therefore, there is a priori no additional numerical diffusion related to the interface handling. For this reason and due to the fact that the SPH approach is well suited to problems with large density contrast, free-surfaces and complex geometries, the SPH method is increasingly used for hydro-engineering and geophysical applications, see [1, 5] for review.

In many situations, due to both: physical processes and numerical errors, the interface in multi-phase flows and the surface in free-surface flows can be fragmented. As a result, the single SPH particle solids/droplets (or even bubbles) can appear and move in the opposite phase or travel without any friction through the regions without particles (representing gaseous phase in free-surface flows). Since the reason of the appearance of single SPH particle solids/droplets may be different depending on the nature and scale of the problem, we do not intend to discuss the causes in detail. In the present paper, however, we focus on the problem of physical correctness of such single particle solids/droplets motion. We try to answer the following questions: is the motion of a single particle droplet physically correct? What is such a droplet’s diameter? What is an influence of the SPH numerical parameters such as $h$, $h/\Delta r$ or the shape of the kernel on the motion of the single particle? Next, we propose two different remedies to the problems related to the fragmentation. The first one, based on adding an artificial repulsive force between the interface, is introduced to minimize the process of fragmentation, for details see [3, 4]. The role of the second one is to filter single particles of one phase moving in the opposite phase.

In the following we assume that the reader is familiar with the SPH basics. Those not familiar with the method may find useful information on the SPHERIC - SPH European Research Interest Community web page [2]. Another important source of information is a handbook by Violeau (2012) [5].

2. Single particle droplets/solids

To illustrate this problem, we consider a multi-phase dam-breaking problem whose evolution is presented in Fig. 1. This simulation involves two liquid columns: black ($B$) and gray ($G$) enclosed in the square box of the edge size $2H$, where $H$ is the height of the black column. The density and viscosity ratios between phases are $\varrho_B/\varrho_G = 4$ and $\mu_B/\mu_G = 2$, respectively. The Reynolds number is $Re = \varrho_B g H^2/\mu_B = 2000$. The simulations were performed for $h/H = 32$, $h/\Delta r = 2$ and the Wendland kernel. Due to gravity, $g$, the denser (black) fluid spreads over the bottom of the tank. Since the tongue of black phase propagates, it induces a violent sloshing flow of the lighter (gray) phase. The presented simulation reveals two events of interface fragmentation whose physical interpretation is not clear. The first one, indicated in Fig. 1 by frame A manifests itself as the appearance of the single particle droplets of the denser phase within the lighter one. The second one, indicated in Fig. 1 by frame B results in fragmentation of the free-surface which leads to the release of particle droplets into the area free of particles, representing an air. From the physical point of view, in both: A and B cases, the single droplets could appear in the flow as an effect of the instabilities in the flow, e.g. Rayleigh-Taylor or Kelvin-Helmholtz in the case A, the Rayleigh-Plateau in the case B, and shear. However, in the case of the SPH modelling, the size of a single droplet is determined by the size of a single SPH particle. Since the single SPH particle is represented by position (point),
Figure 1: Multi-phase dam-breaking problem evolution. Frames A and B indicate regions where some single SPH particle solid/droplets appear and travel through the domain.

density and mass, the definition of the SPH particle size is not obvious. Although the mass of a particle is usually fixed, the density can change which implies the change of the particle volume. Therefore, the physical interpretation of the process of appearance of single droplets is not evident. Probably, in most cases such a behaviour is interpreted as a numerical error related to the low-resolution, sub-kernel particle motion, or/and non-physical pressure pulsation in WCSPH or lack of volume conservation in ISPH. The problem is that a dense droplet swarms as presented in Fig. 1(A) or free-falling droplets, see Fig. 1(B), hitting (with high velocity) the free-surface can significantly modify the flow in the whole domain. Nevertheless, for the sake of the discussion, we will assume that the resolution is high enough to interpret appearance of the single particle droplet as a physically correct process. Then, we will present sets of numerical experiments whereby we will by able to answer all the raised questions.

3. Avoiding the interface fragmentation

The main reason for the interface fragmentation in SPH is the lack of mechanism assuring immiscibility of phases. To prevent this effect and control the interface sharpness, an additional term to the Navier-Stokes equation [4] was introduced

$$\Xi_a = \frac{\varepsilon}{m_a} \sum_{b, c} \left( \frac{1}{\Theta_a} + \frac{1}{\Theta_b} \right) \nabla_a W_{ab}(h),$$

(1)

where summation is done over the neighbouring particles, $m$ is the particle mass, $\Theta$ is the SPH measure of particle number density, $c$ is the color function (used to distinguish phases), $W(h)$ is the Wendland kernel, while $\varepsilon$ is the empirical constant. The role of this term is to add a small repulsion between particles belonging to the opposite phases. The Eq. (1) very efficiently prevents the interface fragmentation, however, in the case of appearance of single particles of one phase in the other, we propose a simple filtering. If $N_a$, defined as

$$N_a = \sum_{b, c} 1,$$

(2)

is smaller than the threshold $N_{\text{min}}$, then we suitably modify the mass, the density and the viscosity coefficient to match the characteristics of the second phase. The usefulness of the proposed remedies was validated by common numerical multiphase tests: the Rayleigh-Taylor instability and the bubble rising in liquid.

References


On the elaboration of a methodology to experimentally verify terminal ballistics models for small arms ammunition

Djalel Eddine Tria, Radosław Trębiński and Jacek Janiszewski

1, 2, 3 Faculty of Mechatronics and Aviation, Military University of Technology
ul. gen. Sylwestra Kaliskiego 2, 00-908 Warsaw, Poland
e-mail: dtria@wat.edu.pl, rkt@wat.edu.pl, jjaniszewski@wat.edu.pl

Abstract

In the present work, experimental and numerical investigations were performed in order to elaborate a reliable methodology for the verification of terminal ballistics models implemented in finite elements (FE) hydrocodes. Ballistic test program was developed where armour steel plates were subjected to 7.62 Armour Piercing projectile at high velocity impact and different test conditions. Numerical simulations with full 3D models for the AP bullet and high strength armour target were carried out in LS-DYNA using Solid Lagrange and Hybrid Solid/SPH elements. The numerical simulations were performed in the same ballistic tests conditions. A measure which characterizes the difference between experimental and modelling results was defined. Because one of the aims of the experimental verification is revealing weak points of models, material tests with novel instrumentation were performed in order to determine material behaviour under quasistatic and dynamic loading conditions. The data were used to calibrate a selected strength and damage/failure models. The methodology showed that stress triaxiality and strain rate based models were found to give results in a good agreement with experimental results and several physical mechanisms are predicted well.

Keywords: Ballistic tests, perforation mechanisms, damage/failure criteria, 3D finite elements simulations

1. Introduction

During the 1990s, and to-date, many hydrocodes have been evolving with powerful numerical algorithms to handle contact, shock wave propagation, large deformation, fracture and fragmentation [1, 5, 6]. The practical application of computer models needs experimental verification [2, 4, 7]. This study aims at developing a methodology to assess if a well calibrated material model is able to predict the eventual physical mechanisms related to high velocity impact and perforation of armour plates by small munitions.

It is foreseen to use results obtained during the methodology in two ways. The obtained experimental results may serve as reference results for analysing various models accessible in commercially available hydrocodes. They can also be used for testing new models, including some analytical or semi-analytical models.

Such a method can be incorporated into code and design guidance for use in military and civil structures to predict effects of these munitions for various conditions of interaction with the structure.

2. Concept of the methodology

i. Material characterization: quasistatic and dynamic tests of projectile and target materials.
ii. Calibration of model parameters
iii. Ballistic tests at well-determined test conditions
iv. Calculations by the use of a model for conditions of ballistic tests.
v. Assessment if a model properly predicts results of experiments on the basis of an analysis of a measure build on a difference between results of experiments and modelling and a measure characterizing random scatter of experimental results.

The measure used in the item “v” is a difference between mean value of given parameter $P_i$ and the value predicted by the use of a theoretical model $P_i^t$, related to the sample standard deviation $S_i$:

$$z_i = \frac{P_i - P_i^t}{S_i}$$  \hspace{1cm} (1)

3. Material behaviour and modelling

A material test program has been carried out on 30PM armour steel plates. The effects of high strain rate and stress triaxiality state on the strength and the ductility of the material were determined. The obtained curves were used to calibrate the MJC strength and failure models, the Cockcroft and Latham (CL) failure models and to determine the effective failure strain and the maximum shear stress at failure.

4. Ballistic tests

A serial of ballistic tests were performed where the 30PM steel plates are subjected to 7.62 AP projectiles at different interaction conditions. The armour plates of different thickness (thin, intermediate and thick) are subjected to normal and oblique impact and fundamental parameters which are used in the comparison with modelling results were measured during and after the shot tests with novel instruments under well-controlled conditions.

5. Numerical simulations

Numerical simulations with full 3D models for the AP bullet and armour plates were carried out in LS-DYNA [3]
using Solid Lagrange and Hybrid Solid/SPH elements. Numerical simulations were performed in the same ballistic test conditions. Figure 1 shows parameters compared to experimental tests.

6. Comparison and verification

In order to represent graphically results of applying the methodology, the Student – $t$ distribution is used. Figure 2 shows this distribution for the number of degrees of freedom equal 6. Vertical lines correspond to the values of the measure $z_i$. Positions of these lines in relation to the distribution illustrate how well a given theoretical model approximates results of experiments.

7. Concluding remarks

- A new methodology to verify experimentally the terminal ballistics models for small arms ammunition was developed.
- Experimental data on the penetration and perforation of amour plates by small arms ammunition were obtained.
- A method for a full 3D modelling of high velocity impact was presented where an adaptive Solid/SPH algorithm was used to handle high element distortion.
- Stress triaxiality and strain rate based models found to give results in a good agreement with experimental results and several physical mechanisms were predicted well.

References


The method of fundamental solutions with optimization of source intensities approach

Tomasz Walczak¹, Grażyna Sypniewska-Kamińska²

¹,² Institute of Applied Mechanics, Poznan University of Technology
Jana Pawła 24, 60-963 Poznań, Poland

e-mail: tomasz.walczak@put.poznan.pl, grażyna.sypniewska-kaminska@put.poznan.pl

Abstract

The paper deals with modification of the method of fundamental solutions (MFS). The classical approach in MFS leads to a system of algebraic linear equations system with unknown intensities of fundamental solutions. A functional attitude is proposed in order to minimize the error of the method. A functional is introduced in order to minimize the values of the investigated source coefficients. This attitude makes it possible that the number of source points may be higher than the number of collocation points. The errors values are smaller than in the case of a simple collocation method. Calculations were carried out for both the Dirichlet boundary conditions and the Neumann conditions. The influence of chosen parameters such as the numbers of the collocation and source points, the distance of the source points from the boundary of the domain, on the values of errors was examined.

Keywords: meshless methods, method of fundamental solutions, source points, collocation

1. Introduction

The method of fundamental solutions is a numerical procedure of a meshless type, designed to solve boundary value problems. Its main idea consists in using the fundamental solutions of the differential equations as the basic functions for the solution of the boundary problem [1-5]. In order to implement this method and to solve an engineering problem, two sets of points should be defined the set of source points outside the considered domain and the set of collocation points on the boundary of a considered body. The accuracy of a numerically obtained solution strongly depends on the number of the collocation points and the number of the source points and on the arrangement of the sources [6,9,10]. Thus, the method of fundamental solutions requires a sufficient amount of source points.

For any given linear boundary value problem the solution \( u(\mathbf{x}) \) of the problem is a linear combination of the fundamental solutions:

\[
  u(\mathbf{x}) = \sum_{k=1}^{N} a_k G_k(\mathbf{x}, \mathbf{z}_k),
\]

where \( \mathbf{x} \in \Omega \cup \partial \Omega \), \( \mathbf{z} \in E^3 - (\Omega \cup \partial \Omega) \), \( \Omega \) denotes a considered body, \( G_k \) are fundamental solutions appropriate for the differential operators of the equations of the considered problem, \( \mathbf{z}_k \) denotes sources and \( a_k \) are unknown coefficients. Strictly, the solution presented in the form of a finite sum is approximate only. Introducing the linear combination (1) of the problem, the system of \( M \) linear equations is obtained with \( N \) unknown coefficients. Two cases are possible:

- the number \( M \) of collocation points is equal to the number \( N \) of source points, then the system of \( N \) linear equations with \( N \) unknown coefficients is represented by a square matrix.
- the number \( M \) of collocation points is higher than the number \( N \) of source points, then the main matrix of the system of linear equations is rectangular and solutions are obtained in least square sense.

A case of a number of the source points greater than the number of the collocation points is also possible. A supplementary requirement, that the values of sources intensities, i.e. unknown coefficients in equation (1), were as small as is possible, seems to be a good solution of such a problem. Such individual approach to the MFS is the aim of the paper.

2. Sources intensities minimization

In order to carry out the calculations with the use of any meshless method, as many collocation and source points as possible should be introduced. This usually leads to an ill-conditioned main matrix of the system. The intensities of the sources achieve huge values then. To avoid such an undesirable situation their values should be minimized.

Let us define the functional \( J \) in the form:

\[
  J(X_1, \ldots, X_M, \alpha_1, \ldots, \alpha_K) = \sum_{i=1}^{N} \sum_{j=1}^{M} |X_j \cdot X_i + \alpha_i (A_i \cdot X_j - B_i)|^2,
\]

where \( X_i \) are unknown coefficients (\( \alpha_i \) in eq. 1), \( \alpha_i \) are additional unknown parameters, \( B_i \) are elements of the right hand-side vector, \( M \) denotes the number of the collocation points and \( N \) denotes the number of the source points.

The minimum of functional \( J \) is investigated:

\[
  \{ X_i \} \in \arg \min \{ J \} \quad \{ \alpha_i \} = \{ \alpha_i \}^* \quad i = 1, \ldots, N
\]

In order to test this approach a number of basic mechanical problems were solved.

3. Numerical example

Let us consider a basic elastostatic problem (presented in [6,7]) to find the displacement field in a steel cube with \( E=200 \) [GPa] and \( \nu=0.3 \). The boundary conditions for each wall are \( u_1=kx_1, u_2=kx_2, u_3=kx_3 \), where \( k=0.0001 \). An exact solutions of

*The work is supported by grant 02/21/DSPB/3463
this problem is known. Approximate solutions of the problem with the use of MFS was widely described in [6-8]. In order to compare the approximate solution of the problem with the exact one, the root mean square error measured on the boundary is defined in the form:

$$\text{Err} = \frac{\sum_{j=1}^{L} \frac{1}{L} \sum_{i=1}^{L} (u_i(P) - \tilde{u}_j(P))^2}{M}$$

(4)

where $L$ denotes the number of control points, $u$ is the exact value of the solution and $\tilde{u}$ is the approximated solution determined in the same point and the set of the test points $P_j$ is included in the boundary of a considered cube.

Numerical simulations for considered problem were done for three cases: the number of collocation points bigger than the number of sources ($M > N$), the number of collocation points equal to the number of sources ($M = N$) and the number of collocation points less than the number of sources ($M < N$). The last case was analysed minimizing the functional $J$ defined in (2).

The values of errors measured on the boundary (4) with respect to distance between collocation points and source points are presented in Fig. 1.

![Figure 1: Err with respect to distance $d$ of sources from boundary, measured in percentage of basic cube dimension.](image)

Note, that the sources were located on a similar cube to the given one throughout the whole simulation.

The proposed approach is the most effective all considered cases for a distance $d$ is less than 300% of a basic cube dimension. The lowest accuracy of an approximate solution is achieved, when the number of collocation points $M$ is greater than the number of source points $N$.

The minimization of source intensities allows to improve the accuracy of approximation. For most numerical simulations obtained error measured on the boundary of a considered body, was lower than in the classical approach. The only disadvantage of a proposed variant of the MFS is significantly bigger size of the main matrix of the system, what enlarges the time of computations.

References


Landslide modelling by the material point method

Zdzisław Więckowski
Faculty of Civil, Architectural and Environmental Engineering, Łódź University of Technology
Al. Politechniki 6, 90-924 Łódź, Poland
e-mail: zwi@p.lodz.pl

Abstract

A three-dimensional landslide problem is analysed by use of the material point method (MPM). MPM is a variant of the finite element method formulated in an arbitrary Lagrangian–Eulerian description of motion. It can also be regarded as a point based method as it traces the state variables at the material points which are defined independently of the computational mesh of the Eulerian type. The method allows to analyse problems with large deformations which usually occur in landslide phenomena. The soil is treated as a cohesive–granular material and described by an elastic–viscoplastic constitutive model with the Drucker–Prager yield condition and a non-associated flow rule. A numerical example related to a three-dimensional problem is included.

Keywords: landslide, granular flow, material point method, arbitrary Lagrangian–Eulerian formulation, 3D analysis

1. Introduction

Modelling of landslides is an interesting task for engineers as the estimation of displacements and velocities of moving soil is important. The study of these fields allows to predict the risk and range of the phenomenon. Large deformations and large strains are involved in the analysis of landslides as significant soil distortions are observed during such processes. This means that computational tools usually used in engineering, e.g. the finite element method (FEM), are not sufficiently robust in this case and more efficient techniques should be developed.

The considered problem is analysed by use of the material point method (MPM) which was successfully applied to two-dimensional large deformation problems of handling granular materials like discharging and filling silos and some problems of geomechanics [4]. In contrast to other frequently used approaches like SPH (smooth particle hydrodynamics) and DEM (discrete element method), MPM has some advantages related to direct enforcing kinematic boundary conditions and the possibility of using well-developed constitutive relations as in the case of standard FEM. The method, well-known in fluid mechanics and other branches of computational physics as the particle-in-cell method, was introduced by Harlow [1]. In the beginning of nineties of the previous century, the method was adapted to problems of solid mechanics, e.g. [3]. MPM, which is an arbitrary Lagrangian–Eulerian formulation of the finite element method, allows one to model large strain problems as it traces the motion of particles with respect to an Eulerian mesh which can be arbitrarily defined. The elastic–viscoplastic constitutive model is applied in the present paper to describe the mechanical behaviour of the soil.

2. Problem description

A dynamic large deformation problem is analysed. The solution of the problem satisfies the following variational equation:

$$\int_{\Omega} \left[ (g a_i \delta v_i + \sigma_{ij} \delta v_{i,j}) \right] d\Omega = \int_{\Omega} g b_i \delta v_i \, d\Omega + \int_{\Gamma_s} t_i \delta v_i \, ds \quad \forall \delta v_i \in V_0$$

and the initial conditions

$$v_i(0) = 0, \quad \sigma_{ij}(0) = \sigma_{ij}^0 \quad \text{on } \Omega$$

where \(V_0\) denotes the space of kinematically admissible velocity fields and \(\Gamma_s\) the part of boundary of region \(\Omega\) where tractions are given. In Equation (1), \(g\) is the mass density, \(a_i\) the acceleration vector, \(v_i\) the velocity vector, \(\sigma_{ij}\) the Cauchy stress tensor, \(t_i\) the vector of volumetric forces, and \(f_i\) the Cauchy stress vector. Symbol \(\sigma_{ij}^0\) in Equation (2) denotes the initial stress field being caused by gravity forces and found by solving an appropriate quasi-static problem.

The soil is modelled by use of the elastic–viscoplastic constitutive relations with the Drucker–Prager yield condition and a non-associated flow rule implying the plastic incompressibility of the material [4]. Let \(f\) denote the yield function \(f(\sigma_{ij}) = q - m p - k\), where \(m = 18 \sin \varphi/(9 - \sin^2 \varphi)\) and \(k = 18 \cos \varphi/(9 - \sin^2 \varphi)\) are functions of the angle of internal friction, \(\varphi\), and cohesion \(c\), \(p\) and \(q\) are invariants of the stress tensor, \(p = -\frac{1}{3} \sigma_{ii}\), \(q = \sqrt{\frac{3}{2} s_{ij} s_{ij}}\), where \(s_{ij} = \sigma_{ij}^e + p \delta_{ij}\) denotes the deviatoric part of the stress tensor. The constitutive relations for the elastic–viscoplastic model are as follows:

$$\dot{p} = -K d_{kk}, \quad e_{ij} = e_{ij}^v + e_{ij}^p, \quad e_{ij}^v = \frac{1}{2G} s_{ij}, \quad e_{ij}^p = \gamma (\Phi(f)) \frac{\partial g}{\partial s_{ij}}$$

where \(g\) denotes the plastic potential defined by relation \(q = g\). The following notation is used above: \(d_{ij} = \frac{1}{2} (v_{ij} + v_{ji})\) is the rate-of-deformation tensor, \(e_{ij}^v\) and \(e_{ij}^p\) are parts of its deviator, \(e_{ij} = d_{ij} - \frac{1}{2} d_{kk} d_{ij}\), the elastic and plastic ones, respectively, \(\sigma_{ij} = \sigma_{ij}^e - \sigma_{ik} \omega_{kj} - \sigma_{ik} \omega_{ki}\) is the Zaremba–Jaumann rate of the stress tensor, \(\omega_{ij} = \frac{1}{2} (v_{ij} + v_{ji})\) the spin, \(K\) and \(G\) are the bulk and shear moduli, respectively. Symbol \(\gamma\) in Equation (3) denotes the viscosity parameter while the function defining the law of plastic flow has the following form:

$$\Phi(f(\sigma_{ij})) = \left( \frac{q - m p - k}{m p + k} \right)^N$$

where \(N > 0\), and

$$\langle \Phi(f) \rangle = \begin{cases} \Phi(f) & \text{if } f > 0 \\ 0 & \text{if } f \leq 0 \end{cases}$$

197
3. Material point solution

The three-dimensional implementation of MPM applied in the paper is an enhancement of the two-dimensional approaches described in [3] and [4]. Two kinds of spacial discretisation are used in MPM. The Lagrangian discretisation is done by dividing the initial configuration of the analysed body into a set of subregions represented by material points. With this discretisation, the mass density field can be expressed by masses of the points $M_P$, and the Dirac delta-function as follows:

$$\phi(x) = \sum_{P=1}^{N} M_P \delta(x - X_P),$$

where $X_P$ denotes the position of the $P$th material point. Another kind of the spacial discretisation is based on an Eulerian finite element mesh (called a computational mesh) covering the virtual position of the analysed body. This mesh can be changed arbitrarily during calculations or remain constant. After substituting Equation (4) to the equation of virtual work (1) and expressing the field of acceleration, $a_i$, and velocity, $v_i$, by the shape functions and nodal parameters defined on the computational mesh as in FEM, the following system of dynamic equations is obtained:

$$M \ddot{X}_i = P_\text{ext} - P_\text{int} + \frac{1}{\rho} \int_{V} \mathbf{F} \cdot \delta(x - X_P) \, dV$$

where $M$ is the mass matrix, $a_i$ the vector of nodal accelerations, $P_\text{ext}$ and $P_\text{int}$ are the vectors of external and internal nodal forces, respectively. The main difference between the finite element and material point methods is based on the fact that the state variables are traced at the material points, defined independently of the computational mesh in MPM, and at integration points connected with elements in FEM. In the three-dimensional analysis, the tetrahedral element with the linear interpolation functions is implemented. The system of equations (5) is solved by means of an explicit time integration procedure with the diagonalised mass matrix.

To avoid solving the quasi-static problem with large stiffness matrix in order to calculate the initial stress field, $\sigma_{ij}^0$, the dynamic relaxation method is applied by setting a small value for viscosity parameter $\gamma$ in the constitutive model. Furthermore, to damp the vibration related to volumetric strain, the following term is added to the pressure: $p_i = -\mu \frac{\partial \varepsilon}{\partial t} (\mu = \text{const}).$

4. Example

A landslide of a soil dike, the cross-section of which is shown in Figure 1, is analysed. It is assumed that the computational region has length of 20 m and a bottom layer with depth of 4 m is added to the region. Due to symmetry, only one half of the region is considered in the computations. It is assumed that the landslide is caused by reducing the material parameters of the soil in the hatched region shown in Figure 1 which can be a result of some physical processes or animals activity.

![Figure 1: Analysed cross-section of a dike, dimensions in metres](image)

Computations have been made with the following material data: $\varrho = 1800$ kg/m$^3$, $E = 1 \cdot 10^7$ Pa, $\nu = 0.3$, $\varphi = 30^\circ$ ($\varphi = 5^\circ$ in the degraded area), $c = 0$ Pa, $\gamma = 200$ s$^{-1}$, $N = 1$. The degraded volume is a part of an ellipsoid with half-axes of dimensions 7, 3 and 4 m. The initial displacement and stress fields have been found by use of the dynamic relaxation technique with the viscosity parameters $\gamma = 5$ s$^{-1}$ and $\mu = 1 \cdot 10^5$ Pa·s.

The computational mesh has been generated using program NETGEN [2]; 15927 nodes and 73000 tetrahedral elements have been defined. The soil material has been represented by 468184 material points. The time increment applied in the explicit algorithm of integration of equations of motion has been set to value $1 \cdot 10^{-3}$ s for time less than 3 s and $0.5 \cdot 10^{-3}$ s for later instants.

The initial configuration and some deformation phases of the soil are shown in Figure 2. Large deformation and large strains of the material are observed. The deformation pattern on the symmetry plane is shown; the shape of the slip line—similar to that observed in experiments—can be recognized in the figure. The pattern of deformation of the dike surface is also visible. It is seen that the points located at a distance greater than about 7 m from the symmetry plane remain at their initial position. The calculations show that, after collapsing, the material achieves its new equilibrium state in 7 s.

![Figure 2: Selected stages of the landslide process: initial position (top left), and positions for 1 s (top right), 2 s (bottom left), 4 s (bottom right)](image)

Although the explicit technique is used to solve the equations of motion, the computation time is not excessively long. The computations (with the use of a computer equipped with Intel Xeon CPU, 3.6 GHz) took 1.18 and 5.12 hours for the quasi-static and dynamic analyses, respectively.

References


Mathematical Methods in Solid Mechanics, Biomechanics and Optimization
– a Session in Honor of Prof. Joachim Telega in the 10th Anniversary of His Death

organized by T. Lewiński and B. Gambin
Viscoelastic–Viscoplastic Material Model for Nonlinear Deformation of Dental Resin Composites

Olurotimi Adeleye¹, Omotayo Fakinlede², Joseph Ajiboye³, Cyril Adegbulugbe⁴

¹,² Department of Systems Engineering, University of Lagos, Lagos, Nigeria
e-mail: oadelaye@unilag.edu.ng, oafak@unilag.edu.ng
³, Department of Mechanical Engineering, University of Lagos, Lagos, Nigeria
e-mail: josyboye@gmail.com
⁴ Department of Restorative Dentistry, College of Medicine of the University of Lagos, IIdiaraba, Lagos, Nigeria

Abstract

The rate-dependence, recoverable and irrecoverable nonlinear deformation behaviour of microhybrid and nanofilled dental resin composites cured with the conventional light emitting diode (LED) and exponential light emitting diode (HiLED) composites under uniaxial loading is here presented. Experimental study based on displacement controlled multiple stress creep recovery (MSCR) tests revealed hysteresis loop in loading and unloading states indicating an inherent viscoelastic nature, and at stress values less than 4 MPa, rate dependent recoverable strains were observed. But at stress values higher than 4 MPa, irreversible strains were observed which also indicates viscoplasticity. Hence a viscoelastic-viscoplastic constitutive model can be used to represent the material deformation behaviour for the dental resin composite.

Keywords: Dental Resin Composites, Deformation Behaviour, Multiple Stress Creep Recovery, Viscoelastic – Viscoplastic

1. Introduction

The usage of polymers and composites in various fields of application has progressed significantly over the past decades. However, in spite of the superior properties of these materials, their use in critical load bearing members is still limited. One of the main reasons for this limited application is the difficulty in reliable prediction of the deformation behaviour of the materials [1]. Most polymeric materials exhibit rate-dependent linear behaviour, but at high temperature, sustained loading or high stress level, these materials undergo nonlinear deformation behaviour. Polymeric composites form heterogeneous systems with distinct microstructural geometries and constituent properties. The heterogeneity in the composites driven by the different properties of the constituents and various microstructural arrangements makes the prediction of time-dependent recoverable and irrecoverable behaviour of composites more complicated [2].

Dental resin composites are materials widely used in restorative dentistry. Besides acceptable aesthetics properties, they can be directly bonded to tooth structure without removing healthy tissues. Because of its bond ability, they can be used for different purposes such as: anterior and posterior teeth injured or diseased by the caries process, occlusion adjustments, different purposes such as: anterior and posterior teeth injured or fractured, they can be directly bonded to tooth structure without removing any healthy tooth material. The study showed that the dentin is mechanically isotropic high elastic and strong hard tissue, which demonstrates considerable plasticity and ability to suppress a crack growth [4].

Viscohyperelastic model has been used to describe the shrinkage of dental resin composite [5]. The proposition in the study is based on the Maxwell model, in which the Young’s modulus and viscosity were continuous functions of time. A viscoplastic model for the shrinkage of dental resin composites due to curing has been developed and solved with FEM program [6]. The implementation of a viscoelastic model with cure – temperature – time superposition principle into the COMSOL Multiphysics software was done [7]. The base equations in the software were modified for the implementation. Although a few studies have been done to identify deformation behaviour of dental resin composites due to shrinkage, its deformation behaviour under uniaxial loadings has not been properly characterized. Hence, the aim of this study is to determine the deformation behaviour model for the resin based microhybrid and nanofilled composites under uniaxial loading.

2. Characterization of the material behaviour

2.1. Materials and Specimen Preparation

Two commercially available particle reinforced resin based dental restorative composites, Filtek Z250 A3 Compules, a universal microhybrid restorative, 20-20 Gm. compules which contains 60 vol% zirconia/silica fillers with particle size ranging from 0.01 to 3.5 μm (average 0.6 μm) and matrix composition of Bisphenol A-Glycidyl Dimethacrylate (Bis-GMA), Bisphenol A-Ethoxylate Dimethacrylate (Bis-EMA), Urethane Dimethacrylate (UDMA) and Triethyleneglycoldimethacrylate (TEGDMA), while the other is nanofilled light cured Universal Fine Hybrid Nano Composite 4g, VITA A1 which contains inorganic fillers; barium glass, silicon dioxide and mixed oxide with the fillers’ particle size between 40 nm and 3 μm. The total content of fillers is 61 vol% while its matrix composition includes Bis-GMA, Urethane Dimethacrylate with 18.8% total monomer content of TEGDMA, Urethane Dimethacrylate (UDMA) and Triethyleneglycoldimethacrylate (TEGDMA).
2.2. Testing and characterization of the material behaviour

Sixty samples of rectangular bar shaped specimens of the two resin based dental restorative composites placed into four groups according to curing modes were prepared for testing using an aluminum split molds of dimensions 2mm X 2.5mm X 8mm as specified by a manufacturer instruction. Composites filled molds were illuminated for polymerization with the light curing units; Flash max2, cms, Light Emitting Diode, Dental Aps, Copenhagen, Denmark, which emits 2400 mW/cm² light intensity exponentially curing for five seconds and Flashlite 2.0, Light Emitting Diode denmat which emits 900 mW/cm² light intensity conventionally curing for twenty seconds.

The MSCR Test for deformation behaviour was done on the BOSE® ElectoForce (ELF) 3200 testing machine. The test was done in a temperature controlled environment at 300C in accordance with ASTM standard method (ASTM D7405 - 08). Ten loading – unloading cycles were applied in the test with the following stress level 5, 10, 15, 20, 25, 30, 35, 40, 45, 50 MPa. The test applies step loading where one load cycle is comprised of 1.0 second of loading followed by 9.0 seconds of unloading within each cycle. A second test to determine path dependency was done. It consists loading and unloading process without increasing the loads in stepwise order.

3. Results and discussion

Figures 1a-d show the results of the MSCR test for microhybrid and nanofilled cured with HiLED and LED. These results showed that, the step loads are followed by the unloading period with the unloading returning to zero strains within the first eighteen seconds for microhybrid and first five seconds for nanofilled. This is time dependent reversible behaviour known as viscoelastic. After this period of eighteen seconds for microhybrid and five seconds for nanofilled, the strains did not fully return to zero strains. In addition, there was gradual increase in the irreversible these strains. These accumulated irreversible strains is referred to as the viscoplastic strains.

Figures 2 a-d shows the stress obtained in the MSCR tests. It can be shown that, after eighteen seconds for microhybrid and five seconds for nanofilled, the viscoplastic strains detected in the tests occurred at a load of 3.5-4.5 MPa. Figure 3 shows the hysteresis loop observed in the deformation of the material which indicates viscoelasticity. This further confirms the path dependent behaviour of the dental resin composites during deformation.

![Figure 1: MSCR test for total strain versus time for (a) microhybrid cured with HiLED (b) microhybrid cured with LED (c) nanofilled cured with HiLED (d) nanofilled cured with LED](image)

![Figure 2: MSCR test for stress versus time (a) microhybrid Cured with HiLED (b) microhybrid cured with LED (c) nanofilled Cured with HiLED (d) nanofilled cured with LED](image)

![Figure 3: Hysteresis loop in mycrohybrid cured with (a) LED and (b) HiLED](image)

4. Conclusion

Analysis of experimental study through MSCR test led to the identification of one distinct regime of viscoelastic behaviour and another regime of viscoplastic behaviour in the deformation behaviour of dental resin composite. At stress values less than 4 MPa, viscoelastic behavior was observed in all the samples, but at stress values beyond 4 MPa, viscoplastic behavior was observed and these two types of deformation behavior were effectively separated. Hence a viscoelastic-viscoplastic constitutive model can be used to represent the material deformation behavior for the dental resin composite.

References


Analytical solution and numerical simulation of borehole ground heat exchangers for geothermal heat pump systems: ground influence zone

Piotr Alawdin*; Jakub Marcinowski*

1,2 Faculty of Civil Engineering, Architecture and Environmental Engineering, University of Zielona Góra
prof. Z. Szafrański 1, 65-516 Zielona Góra, Poland
e-mail: P.Alawdin@ib.uz.zgora.pl 1, J.Marcinowski@ib.uz.zgora.pl 2

Abstract

A zone of influence of individual vertical borehole in real ground is identified in analytical and numerical calculations. The inhomogeneous layered medium is taken as a mathematical model of problem for the ground, whose thermal field is described by a Fourier-Kirchhoff equation. The proposed approach can be applied to other, more complex models of soil. Finite element calculations of the system "borehole plus layered soil" as a whole were performed, and verification of the calculations for the individual layers using the program Mathematica was also carried out. An analytical dependence of zone of influence from the thermal parameters of ground layers was obtained, which agreeing well with the results of the finite element calculations.

Keywords: borehole heat exchangers, geothermal pump, analytical and numerical simulation, ground influence zone, layers inhomogeneity

1. Introduction

One of important sources of renewable energy is geothermal heat pump system with boreholes, which are usually placed in groups (Ref. [1]). To design the location of these boreholes it is necessary to clarify the mutual influence of boreholes on the surrounding real soils.

Known publications on the subject used the concept of "zone of influence" of individual boreholes located in homogeneous grounds, but does not contain specific criteria for determining this zones of influence (Ref. [2,3,4]). In other works such a zone of influence (or the adiabatic surface) is given without any justification.

The aim of the paper is to identify and to confirm numerically a zone of influence of individual vertical boreholes in real layered soils. As a mathematical model of problem for the ground the inhomogeneous layered medium is taken, whose thermal field is described by Fourier-Kirchhoff equation. The proposed approach can be applied to other, more complex models of soil, in which the following, given beneath assumptions are satisfied.

Finite element calculations of the system "borehole-layered soil" as a whole (by the system COSMOS/M) were performed. The verification of calculations for the individual layers using the program Wolfram Mathematica was also carried out. As a result, analytical dependence of zone of influence from the thermal parameters of ground layers was obtained, agreeing well with results of finite element calculations.

2. Mathematical model of the problem

2.1. Governing equation and assumptions

A two-dimensional problem of borehole ground heat exchanger is examined numerically for the axially symmetrical geothermal heat pump system (Fig. 1). The ground in a ring volume is a layered isotropic medium and system of axial cylindrical coordinates 0rz is used. The governing differential equation for heat transfer for the nonhomogeneous body is as follows:

\[ \rho \frac{\partial (cT)}{\partial t} = \frac{1}{r} \frac{\partial}{\partial r} \left( r k \frac{\partial T}{\partial r} \right) + \frac{\partial}{\partial z} \left( k \frac{\partial T}{\partial z} \right) \]

where \( T(x, t) \) is a temperature; \( x = (r, z) \) a vector of the body point, \( x \in \Omega \); \( \Omega \) is a cylinder region for variables \( x \in \Omega \subseteq R^2 \); \( t \) is time; \( \alpha(x) \) the thermal diffusivity of ground as a function of coordinate \( x \); \( \alpha = \kappa/(\rho \ c \ s) \); \( k \) is thermal conductivity; \( \rho \) density of the material; \( c \) specific heat capacity for constant pressure. Coefficient \( \alpha \) is discontinuous for the horizontal axis of the cylindrical coordinate, \( z = 0 \), \( a = \{a_1, z < 0 \mid a_2, z > 0\} \). The same analysis was made also for the ground with layers in vertical direction.

A ring volume \( \Omega \) has radii \( r_n, R^* \) of borehole and outward ring dimension respectively. \( R^* \) is really infinite, but for numerical analysis it may be approximated by certain large finite domain. We assume it more than 1.5R, \( R^*>1.5R \), where \( R \) is sought-for radius of influence zone (see p. 2.2).

We introduce the following dimensionless quantities, coordinate and parameters: radius \( r = r/\rho_0 \); influence zone radius ; Fourier number \( F_0 \) (dimensionless time \( \tau = \alpha/r^2 \)); Below all tildes are omitted.

Further, the temperature of the body satisfies the initial condition

\[ T(r, z, 0) = T_a(z) \text{ for } t = 0, \]

as well as boundary conditions:

\[ \frac{\partial T(R^*, z, t)}{\partial r} = 0 \text{ for } r = R^*, \]

\[ T(r_0, z, t) = T_0 + (T_e - T_0) e^{-1000t} \text{ for } r = r_0, \]

where \( R^* \) is unknown value, \( R^*>1.5R \); \( R \) is a root of Eqn (5); \( T_0, T_e \) are given functions of \( z \).

*The authors acknowledge the support and advice from Prof. Zygmunt Lipnicki.
Table 1: Numerical and analytical results of analysis

<table>
<thead>
<tr>
<th>Times $t$ in months</th>
<th>1st layer, Dry soil</th>
<th>2nd layer, Sandstone</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Numerical</td>
<td>Analytical</td>
</tr>
<tr>
<td>1</td>
<td>4.75 m</td>
<td>4.76 m</td>
</tr>
<tr>
<td>1.5</td>
<td>5.70 m</td>
<td>5.72 m</td>
</tr>
<tr>
<td>2</td>
<td>6.70 m</td>
<td>6.75 m</td>
</tr>
</tbody>
</table>

Figure 1: Scheme of borehole in ground with $i, j$ layers and influence zone radius $R$

For further numerical and analytical calculations of the field $T(r, z)$ of the layered medium temperature (Fig. 1) we assume:
1. The infinite branch of temperature curve $T(r)$ in the middle of each $j$-th layer $z = h_j/2$ is an asymptote to the horizontal line, $T(r) = T_o$, or for $r \to \infty$ is hold $T(r) \to T_o$.
2. The derivative of curve $T(z)$ at $z = h_j/2$ is equal to zero, $\partial T / \partial z = 0$.

2.2. Criterion of influence zone

The parameter $R$ of influence zone may be define as a certain point $r = R$ of curve $T(r)$, for which $\frac{\partial^2 T}{\partial R^2} \leq \varepsilon_d$, or in a slightly different form

$$(T_r - T_o(R))/(T_r - T_o) \leq \varepsilon,$$  \hspace{1cm} (5)

where $\varepsilon_d$, $\varepsilon$ are small given values as accuracy of influence zone definition; $T_o$ is temperature of influence zone boundary (Fig. 1).

In that way, the dimension $R$ is a root of a nonlinear Eqn (5), which usually has algorithmic nature.

3. Analytical solution and numerical simulations

Numerical simulations were performed for the temperature fields in "borehole-layered soil" system in homogeneous and nonhomogeneous grounds. Finite element calculations of the whole system "borehole-layered soil" were made using program COSMOS/M. The verification of the calculations for the certain layers was carried out by program Wolfram Mathematica.

We assumed the accuracy of influence zone $\varepsilon=0.1\%$. On the basis of calculations we obtained an analytical approximation of dimensionless influence zone radius $R$ for the certain layer vs reduced Fourier number (dimensionless time) $x=\text{Fo}/\text{Fo}_1$ in the interval $x=(1, 6)$ as follows:

$R=45.4 + (5.28 + (-0.640 + (0.110 + (0.0233 - 0.0133 (-2 + x)) (-5 + x)) (-3 + x)) (1 + x)) (-6 + x)$,

where $\text{Fo}_1=18.3141$. The curve $R(x)$ resulting from Eqn. (6) is shown in Figure (in full text).

3.1. Example 1

A borehole with radius 0.25 m is made in homogeneous dry soil: $k_s=0.58\, \text{W/(mK)}$; $c_s = 796\, \text{J/(kg K)}$; $\rho_s = 1650\, \text{kg/m}^3$; $a_s = 4.41602*10^{-7}\,\text{m}^2/\text{s}$; time interval $t=(1, 6)$ months.

Therefore the interval of values $x=(1, 6)$ and accordingly the zone of influence of the borehole (from Eqn (6)) is in the range of $(4.75, 11.45)\,\text{m}$.

3.2. Example 2

A borehole with radius 0.25 m is in two-layered ground. First layer is dry soil as in Example 1. Second layer is sandstone: $k_s = 1.9\, \text{W/(mK)}$; $c_s = 710\, \text{J/(kg K)}$; $\rho_s = 2250\, \text{kg/m}^3$; $a_s = 11.893*10^{-7}\,\text{m}^2/\text{s}$; time interval $t=(1, 2)$ months.

The zones of influence of the borehole obtained from numerical and analytical (Eqn (6)) calculations are shown in the Table 1.

4. Conclusions

A new criterion and method of calculations of ground influence zone for vertical borehole was suggested. This zone depends on the reduced Fourier number (dimensionless time) for the homogeneous medium.

For the inhomogeneous one (for the ground with layers in vertical direction) it depends also on the ratio of layers thermal diffusivity.

The proposed approach may be useful for any similar or more complex models of real ground.

References


Finite element stress analysis of lumbar vertebrae body during osteoporotic degradation

Oleg Ardatov¹, Algirdas Maknickas², Vidmantas Alekna³, Rimantas Kačianauskas⁴

¹Department of Biomechanics, Vilnius Gediminas Technical University
Basanavičiaus 28, 03224 Vilnius, Lithuania
e-mail: oleg.ardatov@vgtu.lt
²,⁴Institute of Mechanics, Vilnius Gediminas Technical University
Basanavičiaus 28, 03224 Vilnius, Lithuania
³Department of Rehabilitation, Physical and Sports Medicine, Vilnius University
Čiurlionio 21, 03101 Vilnius, Lithuania

Abstract

The study presents results of the nonlinear finite element (FEM) analysis of osteoporotic human lumbar vertebrae L1. The lumbar body consists of two basic parts – inner cancellous bone and outer cortical shell with anatomical geometry which is modelled as inhomogeneous elastic orthotropic body. The osteoporotic degradation of the bone tissue is modelled by reduction of the thickness of cortical shell and the power-law equations. It is subjected by physiological static and dynamic load. The modelling results are presented in terms of stresses which occur on the cortical shell. The results of the nonlinear interdependence between stress and ratio of degradation are shown. The presented results can be useful for both medical diagnosis and bone health check.

Keywords: nonlinear analysis, finite element method, bone tissue elasticity, lumbar vertebrae, osteoporosis

1. Introduction

Mechanical analysis for modelling of biological tissues is recently widely used for solving theoretical problems and in medical practice. Osteoporosis is one of the most common health problems affecting both men and women, and it is becoming increasingly prevalent in our aging society [1,2], therefore, employing of stress results for medical diagnostics needs could be beneficial. The developed FEM model of lumbar body is aimed to serve as a tool to describe osteoporotic degradation of the bone.

2. Problem formulation and initial data

The three-dimensional nonlinear static and dynamic FEM analysis was performed in order to define the mechanical behaviour of human lumbar vertebrae L1 model under the compression load. The inhomogeneous lumbar vertebrae body consists of two basic structural members – outer cortical shell fulfilled by inner bone tissue. The initial anatomical geometry of the developed vertebrae body model is illustrated in Fig. 1A. The height of the model is equal to 30 mm, the cross-sectional size is equal to 40 mm. Width of cortical shell depends on degree of osteoporosis. In this research we are investigating three models with width of 0.5; 0.4 and 0.2 mm. Two intervertebral disks of thickness 10 mm were also included to reflect boundary conditions with the neighbour trabecular.

Both cancellous trabecular bone and surrounding compact bone (cortical shell) are modelled as elastic transversally orthotropic continuum. Intervertebral disks were assumed isotropic and perfectly elastic. Mechanical properties of model members are presented in Table 1.

Osteoporotic influence for the vertebrae is characterised by decreasing modulus of elasticity of cancellous bone. Modulus of cancellous bone is determined according to power-law equations, which reflect the impact of apparent density. The selection of this equation is based on alignment of our research and data published by [3].

\[ E_{cancellous} = 4.730 \rho^{1.56} \]  

There \( \rho \) – apparent density. In current research it is in range between 0.10 and 0.30 g/cm³.

### Table 1: Elasticity constants

<table>
<thead>
<tr>
<th>Part of the model</th>
<th>( E_x ), MPa</th>
<th>( E_y = E_z ), MPa</th>
<th>( \nu_x = \nu_y ), ( \nu_z )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cortical shell</td>
<td>8000</td>
<td>2500</td>
<td>0.30</td>
</tr>
<tr>
<td>Cancellous bone</td>
<td>130-720</td>
<td>42-240</td>
<td>0.30</td>
</tr>
<tr>
<td>Intervertebral disk</td>
<td>10</td>
<td>10</td>
<td>0.495</td>
</tr>
</tbody>
</table>

Figure 1: A – Section view of the model, B – Finite element mesh

The bone is subjected by the time-dependent physiological loads, which occur through daily activities. Generally, it presents the axially acting pressure load (Fig. 2). The load depends on the displacement values while its direction during compression changes with the deformed shape of the model.

Time variation of the load is shown in Fig. 3. It is easy to find that in the first stage until 0.15 MPa the loading is low and has static character while in the second stage in the range of 0.15 and 0.75 MPa dynamic behaviour is expected. Consequently, the geometric nonlinearities were verified by solving static and dynamic analysis models.
Figure 2: Schematization of load due to compression test; E1 – cortical shell; E2 – cancellous bone

Figure 3: Variation of dynamic load in time

A model is meshed with tetrahedral volume finite elements. The number of finite elements of cortical shell was 7686. Cancellous bone: 12915 finite elements, intervertebral disks: 3224 finite elements. The model is characterised by 323 274 degrees of freedom. SolidWorks software was used.

3. Numerical results

The Von Mises-Hencky criterion is chosen to predict the failure of the model. The selection of this criterion is based on mechanical properties of the bone, which intend to behave as a ductile material [4]. Von Mises stress criterion is applied on research of stresses, which occur on cortical shell of the model. Strength capacity is 40 MPa [5]. Strength capacity is assumed to be constant, but its thickness is reducing due to osteoporosis effect from 0.5 mm down to 0.2 mm.

The schemes of the distribution of Von Mises stress on cortical shell of the model were obtained due to static and dynamic load is presented in Fig. 4. It shows that the highest von-Mises stresses occurred in the middle of the cortical shell on the front side of the model. These results are in good agreement according to clinical observations [1].

Figure 4: A – Von Mises stress distribution due to static load, B – due to dynamic load

Distribution of stress due to dynamical load is more varied and there is a higher risk of shell overloading. The risk of fracture appears due to 0.45 MPa static loads in case of low density of trabecular bone (0.1 g/cm³). The risk of fracture is notably higher due to dynamical load. It appears due to 0.45 MPa load at 0.2 g/cm³ bone density. It shows that dynamic loads are more dangerous for lumbar body (Fig. 5).

Figure 5: The dependence between von Mises stress, and density of trabecular bone in model with 0.2 mm shell width due to 0.5 MPa load

The obtained results showed, that von Mises stress greatly depends both on density of trabecular bone and width of cortical shell. Also, the dynamical loads cause the concentrators of stress on cortical shell and increase fracture risk for lumbar body. It shows that static analysis does not fully define the risk of fracture for degenerated bone, and in order to define state of bone health of the patient, dynamic analysis should be performed.

4. Conclusion

Created FEM model of lumbar body showed that verification of strength of lumbar body should include estimation of density, thickness of cortical shell and influence of dynamical load. This model can be individualized according to the peculiar properties of patients. The simplified cancellous bone, allows accelerating of the calculations in order to make an urgent diagnosis for the patient.

References

Fast algorithm for flux around closely spaced non-overlapping disks

Olaf Bar
Department of Computer Sciences and Computer Methods, Pedagogical University,
ul. Podchoraży 2, Krakow 30-084, Poland
e-mail: bar@up.krakow.pl

Abstract

The paper is focused on application of the fast algorithm to determine the flux around closely spaced non-overlapping disks on the conductive plane. This method is based on successive approximations applied to functional equations. When the distances between the disks are sufficiently small, convergence of the classical method of images fails numerically. In the talk, the limitations on geometric parameters are described.

Keywords: Non-overlapping disks, Multiply connected domain, Poincaré series

1. Introduction

Fibrous composites can be modeled by unidirectional circular cylinders embedded in the matrix [1, 2]. The electrical or thermal conductivity in the plane perpendicular to fibres can be described by Laplace’s equation. Standard numerical methods (like FEM) refer to calculation of the local fields. In special cases (like a periodic layout) we can use the Weierstrass function to solve this problem. In the paper we focus on the case without symmetry. The method used in this work gives an approximate analytical solution of the discussed problem [4]. This method is based on inversions with respect to the circles which transform the harmonic functions from inside to outside of the disks.

Denote : \( D_k = \{ z \in \mathbb{C} : | z - a_k | < r_k \} \) \( (k = 1, 2, \ldots, N) \).

\[ z^{*}_{(k)} = \frac{r_k^2}{z - a_k} + a_k \]  

\[ u(t) = u_k, \quad | t - a_k | = r_k, \quad k = 1, 2, \ldots, N. \]

This problem can be reduced to the Riemann–Hilbert problem [6]

\[ Re \varphi(t) = u_k, \quad | t - a_k | = r_k, \quad k = 1, 2, \ldots, N, \]  

on the function \( \varphi(t) \) analytic in \( D \).

The exact solution for the flux between two circles is known [7]

\[ \Psi(z) = \frac{1}{z - z_{12}} - \frac{1}{z - z_{21}} \]

where \( \Psi(z) = \varphi'(z) \). The function \( \Psi(z) \) describes the flux for the known difference \( u_1 - u_2 \) and the \( z_{12}, z_{21} \) satisfy the quadratic equation \( z_{1(2)}^* = z_{2(1)}^* \). This function can be used as the zero-th approximation for the fast algorithm.

Define the analytic function [2]:

\[ f_{km}(z) := \sum_{\ell \in J_m, \ell \neq k} \Psi(z; m, \ell)(z), \quad k \in J_m, \]

where \( J_m \) is the complement of \( J_m \cup \{ m \} \) to \( \{ 1, 2, \ldots, n \} \).

The following algorithm can be applied

\[ \psi_k^{(0)}(z) = f_{km}(z), \]

\[ \psi_k^{(p)}(z) = \sum_{m \neq k} \left( \frac{r_m}{z - a_m} \right)^2 \psi_m^{(p-1)}(z^{*}_{(m)}) + f_{km}(z), \quad p = 1, 2, \ldots, \]

The \( p \)-th approximation of the complex flux is calculated by formula

\[ \psi_k^{(p)}(z) = \sum_{m=1}^{n} \left( \frac{r_m}{z - a_m} \right)^2 \psi_m^{(p)}(z^{*}_{(m)}) + \psi_k(z), \quad z \in \mathbb{D}, \]

where \( \psi_k(z) = \sum_{\ell \in J_m} \Psi(z; m, \ell). \)
2. Results

Calculations were made for four circles. Introduce:
- the vector of the circle centers: \([-1, -i, 1.1, 1.2 i]\) (see Fig. 1);
- the radius \(r = \sqrt{2}\). The limit case \(\delta = 1\) yields tangent circles \(D_1\) and \(D_2\).

The accuracy of calculation are associated to the accuracy of the boundary conditions. The procedure to obtain the potential from the flux requires integration of the long algebraic expressions. The analytical integration procedure requires a lot of time and it is possible at most for the 7 iterations. Thus instead of the checking the boundary conditions we check the equivalent conditions [2]:

\[
I\Psi(t) := \text{Im} \left( \frac{t - a_k}{r_k} \psi(t) \right) = 0, \quad |t - a_k| = r_k,
\]

\(k = 1, \ldots, 4\), \hspace{1cm} (9)

Introduce:

\[
I\Psi_{err} = \max(\{|I\Psi(t)|\}),
\]

\(U_{err} = \frac{\max(u(t)) - \min(u(t))}{\max(u(t))} \hspace{1cm} (10)

where \(t \in \partial\Omega\)

The dependence of logarithm of the error on number of iterations is linear.

\[
\ln /LParen1 U err /Minus 5 4 /Minus 3 /Minus 2 /Minus 1 /Minus 0 0\hspace{1cm} (11)
\]

The results presented in the paper show that at the present stage of research, we can perform at most 15 iterations, thus it is the limitation of the method. For four nonoverlapping disks the minimum distance between inclusions is equal to \(\delta = 0.99\). The corresponding relative error holds less than 5%.

Table 1: Calculation for \(\delta = 0.9\)

<table>
<thead>
<tr>
<th>iterations</th>
<th>(I\Psi_{err})</th>
<th>(U_{err})</th>
<th>(\ln /LParen1 U err /Minus 5 4 /Minus 3 /Minus 2 /Minus 1 /Minus 0 0)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.85</td>
<td>0.28</td>
<td>6.67</td>
</tr>
<tr>
<td>2</td>
<td>0.95</td>
<td>0.13</td>
<td>7.28</td>
</tr>
<tr>
<td>3</td>
<td>0.54</td>
<td>0.07</td>
<td>6.91</td>
</tr>
<tr>
<td>4</td>
<td>0.27</td>
<td>0.04</td>
<td>6.92</td>
</tr>
<tr>
<td>5</td>
<td>0.15</td>
<td>0.024</td>
<td>6.49</td>
</tr>
<tr>
<td>6</td>
<td>0.08</td>
<td>0.013</td>
<td>6.24</td>
</tr>
<tr>
<td>7</td>
<td>0.05</td>
<td>0.0081</td>
<td>5.79</td>
</tr>
</tbody>
</table>

When the number of iterations is less than 8, it is possible to compare the boundary values comparing the \(I\Psi(t)\) and \(U(t)\). The dependence of \(\ln/|U_{err}|\) on the number of iteration is linear. Fig. 2 shows this relation for the eight iterations depending on \(\delta\). Although it is impossible to obtain \(U_{err}\) for higher number of iterations, we can use the linear approximation to estimate this error. This prediction for \(U_{err}\) shown on Table 2.

Table 2: Number of iterations on acceptable \(U_{err}\)

| \(I\Psi_{err}\) | \(U_{err}\) | \(\ln|U_{err}|\) |
|-----------------|-------------|----------------|
| 1               | 1.85        | 6.67           |
| 2               | 0.95        | 7.28           |
| 3               | 0.54        | 6.91           |
| 4               | 0.27        | 6.92           |
| 5               | 0.15        | 6.49           |
| 6               | 0.08        | 6.24           |
| 7               | 0.05        | 5.79           |

Table 3: Calculation for \(I\Psi_{err}\)

<table>
<thead>
<tr>
<th>iterations</th>
<th>(\ln /LParen1 U err /Minus 5 4 /Minus 3 /Minus 2 /Minus 1 /Minus 0 0)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.026 0.246 1.76</td>
</tr>
<tr>
<td>9</td>
<td>0.015 0.160 1.20</td>
</tr>
<tr>
<td>10</td>
<td>0.0082 0.111 1.17</td>
</tr>
<tr>
<td>11</td>
<td>0.0049 0.072 0.791</td>
</tr>
<tr>
<td>12</td>
<td>0.0027 0.050 0.779</td>
</tr>
<tr>
<td>13</td>
<td>0.0016 0.032 0.525</td>
</tr>
<tr>
<td>14</td>
<td>0.0009 0.022 0.523</td>
</tr>
<tr>
<td>15</td>
<td>0.0005 0.015 0.351</td>
</tr>
</tbody>
</table>

Figure 2: Dependence of logarithm of the error on number of iterations

The results presented in the paper show that at the present stage of research, we can perform at most 15 iterations, thus it is the limitation of the method. For four nonoverlapping disks the minimum distance between inclusions is equal to \(\delta = 0.99\). The corresponding relative error holds less than 5%.

References

Laminar flow past the bottom with obstacles - from suspension to porous medium

W. Bielski¹, R. Wojnar²

¹ Institute of Geophysics, Polish Academy of Sciences, IGF PAN
Księcia Janusza 64, 01-452 Warszawa
e-mail: wbierski@igf.edu.pl

² Institute of Fundamental Technological Research, PAS
Pawinińskiego 5B, 02-106 Warsaw, Poland
e-mail: rwojnar@ippt.waw.pl

Abstract

We continue our work on the influence of bottom obstacles on the water flow in canals under the gravity forces, [1]. We propose two approaches to solve this important geohydrological problem:
1. If the obstacles are rare, the fluid flooding the obstacles at the bottom is replaced by a layer of different viscous fluid, and regarded as the flow through the porous medium.
2. If the obstacles at bottom are dense, they can be regarded as a porous medium, and the flow of fluid filling this medium can be treated as the flow through the porous medium.

Keywords: Brinkman’s equations; bottom obstacle

1. Brinkman’s suspension and the suspension flow

Let a solute-particle has a volume \( \omega_0 \) and a spherical shape of radius \( R \). Then \( \omega_0 = (4/3) \pi R^3 \). Let us assume that we have \( N \) solute particles, each with the volume \( \omega_0 \). Thus the concentration by volume is \( c_0 = N \omega_0 / V \). If the contributions of the individual particles forming a suspension are considered independent, the Einstein formula for the effective viscosity of the suspension reads \( \eta = \eta_0 (1 + 5 c_0 / 2) \). The formula was derived in the so-called, non-interaction approximation, and the formula corresponds to summation of the viscosity contributions of individual particles. This means that it is valid only for small concentrations.

Brinkman derived an expression for the viscosity of suspensions of finite concentration

\[
\eta = \frac{\eta_0}{(1 - c_0)^{3/2}} \tag{1}
\]

and found that this equation agrees quite well with experimental results until concentration of 40%. The laminar flow of such suspension is described by a stationary Navier-Stokes equation with variable viscosity.

1.1. Darcy’s law

An empirical relation describing the flow of a fluid through a porous mass is Darcy’s equation,

\[
v = \frac{K}{\eta} (-\nabla p + G) \tag{2}
\]

where \( v \) is the rate of flow through a surface element of unit area, \( K \) is the permeability of the porous medium and, \( p \) is the pressure. Moreover, \( G = \rho g \) is the gravitational body force, \( \rho \) is the fluid density and \( g \) is the earth gravity acceleration, \( g \equiv |g| = 9.81 \text{ m/s}^2 \).

1.2. Brinkman’s equation of seepage

What concerns the relation (2), an objection arises that no viscous stress tensor has been defined in relation to it. The viscous shearing stresses acting on a volume element of fluid were neglected; only the damping force of the porous mass \( \eta v/K \) has been retained. H. C. Brinkman [2] modified Darcy’s relation (2) in the following way

\[
v = \frac{K}{\eta} (-\nabla p + G) + \eta' \Delta v \tag{3}
\]

The factor \( \eta' \) in the term \( \eta' \Delta v \) (it is the divergence of the stress tensor) may be different from \( \eta \).

The Darcy’s and Brinkman’s relations can be derived from Navier-Stokes equation using the asymptotic methods.

2. Fluid with the variable viscosity

Let a layer of an incompressible viscous fluid of thickness \( h \) be bounded above by a free surface and below by a fixed plane inclined at an angle \( \alpha \) to the horizontal. Let the density \( \rho \) of the fluid be constant, while its viscosity \( \eta \) be dependent on the position according to a given law. Let us determine the steady flow due to gravity. Let the velocity has only one component \( v = (v, 0, 0) \) with \( v = v(y) \). We take the fixed plane as the \( xz \)-plane, with the \( x \)-axis in the direction of flow and \( y \) pointed upward in direction perpendicular to the free surface. Let the viscosity be function of \( y \) only and we seek a solution depending only on \( y \). The Navier-Stokes system of equations reduces to two equations

\[
\frac{\partial}{\partial y} \left( \eta \frac{\partial v}{\partial y} \right) + G_x = 0 \quad \text{and} \quad \frac{\partial p}{\partial y} + G_y = 0 \tag{4}
\]

Here \( G_x = \rho g \sin \alpha \) and \( G_y = \rho g \cos \alpha \). The shear component is

\[
\sigma_{xy} = v \frac{\partial v}{\partial y} \tag{5}
\]

After integration of the system and accounting for boundary conditions we get

\[
v(y) = g_x \int_0^y \frac{h - \tilde{y}}{\tilde{y}(\tilde{y})} d\tilde{y} \tag{6}
\]
and for $\eta = \text{constant}$

$$v(y) = \frac{1}{\eta} g_x \left( h - \frac{1}{2} y \right)$$  \hspace{1cm} (7)

Figure 1: Flow of the stream composed of two parts

3. Fluid with two components of different viscosity

Consider flow of the fluid composed of two different immiscible parts: upper part for $h_1 < y < h_1$ has the viscosity $\eta_A$ and the lower part for $0 \leq y < h_1$ has the viscosity $\eta_B$, see Fig. 1. The densities of the both parts are the same. We apply the solution (8) and find for $0 \leq y < h_1$ (part B)

$$v = \frac{1}{\eta_B} g_x y \left( h - \frac{1}{2} y \right)$$  \hspace{1cm} (8)

and for $h_1 < y < h_1$ (part A)

$$v = v_0^{(A)} + \frac{1}{\eta_A} g_x \left( h(y - h_1) - \frac{1}{2} (y^2 - h_1^2) \right)$$  \hspace{1cm} (9)

with $v_0^{(A)} = g_x h_1 \left( h - \frac{1}{2} h_1 \right) / \eta_B$. An example of the dependence of velocity $v$ upon the coordinate $y$ is shown in Fig. 2.

4. Flow in two regions: free and porous

We refer to Fig.1 again. Consider flow of the same fluid with the viscosity $\eta$ developing in two different manners: in the upper part $A$ for $h_1 < y < h_1$ the flow is free, and in the lower part $B$ for $0 \leq y < h_1$ we deal with the seepage through a porous medium.

4.1. Free flow in the part A

From the first equation of the system (4) we get

$$v(y) = \frac{g_x}{\eta} \left( \frac{1}{2} y + h \right) y + v_B$$  \hspace{1cm} (10)

This is the solution for Part A, $(h_1 < y < h_1)$, in which the integration constant $v_B$ is unknown, and is to be found from comparison with the solution in the lower part $B$.

4.2. Flow in the lower part $B$

We apply Brinkman’s equation and we get

$$\frac{d^2 v}{dy^2} - \frac{\eta}{\eta^2 K} v + \frac{g_x}{\eta} = 0$$  \hspace{1cm} (11)

whose solution (for Part B, $0 < y < h_1$) is of the form

$$v = b_1 e^{a y} + b_2 e^{-a y} + K^{-1} g_x$$  \hspace{1cm} (12)

where $a = \sqrt{\frac{\eta}{\eta K}}$.

3 constants are to be found. Since the velocity $v$ at the bottom ($y = 0$) should vanish

$$b_1 + b_2 = - K^{-1} g_x$$  \hspace{1cm} (14)

And the powers of $a$ higher than 2 are rejected. First, we observe that in virtue of Eq.(14) the sum $b_1 + b_2 + (K/\eta) g_x$ vanishes and the constant term in (16) is equal zero.

Next, we find the coefficient at $y^2$, and get

$$(b_1 + b_2) \frac{1}{2} a^2 = - \frac{K}{\eta} g_x \frac{a^2}{2} = - \frac{1}{2} g_x$$  \hspace{1cm} (17)

This result is identical with the coefficient at $y^2$ in formula (7), if only $a'$ is replaced by $\eta$.

The coefficient at linear term reads $(b_1 - b_2) a' \frac{2}{\eta}$, and

$$- \frac{h - h_1}{\eta} g_x - a' \frac{K}{\eta} (1 - a h_1) g_x + \frac{K}{\eta} g_x a = \frac{h}{\eta} g_x$$  \hspace{1cm} (18)

If only $a'$ and $\eta$ are identified, one obtains the linear coefficient of flow described by formula (7). Therefore, our solution exhibits proper asymptotic properties for $K \to \infty$.

References


Simulations of random geometric objects on the plane and their applications

Roman Czapla
Department of Computer Sciences and Computer Methods,
Pedagogical University ul. Podchorazych 2 Kraków 30-084 Poland
e-mail: czapla@up.krakow.pl

Abstract

Random geometric structures are discussed in this paper. The results are applied to heterogeneous media in a wide sense, e.g. to bacteria sets. Random distributions are described using the modified Eisenstein-Rayleigh sums (E-R).

Keywords: random inclusions, Eisenstein-Rayleigh sums, composite material, collective behavior of bacteria

1. Introduction

Effective properties of random composites can be described by n-point correlation functions [9]. However, calculation of higher order correlation functions is a difficult computational task. Instead, we apply modified E-R sums (e-e) introduced in [5]. These sums can be regarded probabilistic moments of the correlation functions. Moreover, it is justified in [5] that the effective properties of 2D composites are expanded in series on \(e_m\), i.e. the sum of \(e_m\) can be considered a basic set for the effective properties.

In the present talk, we simulate random sets of slit and circular inclusions that can be considered a basic set of chaotic locations of inclusions. Further, the real locations of bacteria [8] are used to compute the corresponding \(e_m\). The results are compared and it is established that bacteria do not obey the chaotic law. That is a symptom of their collective behavior.

2. Random location of slits and basic sums

2.1. The representative cell

Let \(\omega_1\) and \(\omega_2\) be the pair fundamental translation vectors on the complex plane \(\mathbb{C}\) such that \(\text{Im}(\frac{\omega_1}{\omega_2}) > 0\). We introduce the \((0,0)\)-cell as the parallelogram \(Q_{(0,0)} := \{z = t_1\omega_1 + t_2\omega_2 : -\frac{1}{2} < t_k < \frac{1}{2} \ (k = 1,2)\}\). The lattice \(Q\) consists of the cells \(Q_{(m,n)} := \{z \in \mathbb{C} : z = m\omega_1 + n\omega_2 \in Q_{(0,0)}\}\), where \(m, n \in \mathbb{Z}\). In the case \(\omega_1 = 1\) and \(\omega_2 = i\) the cell \(Q\) becomes a square and the array \(Q\) is called the square lattice. Consider \(N\) non-overlapping slits \(\Gamma_{k}\) of length \(l\) with the centers \(b_k \in Q_{(0,0)}\) and angle of inclination \(\alpha_k \in [0, \pi)\). (see Fig. 1).

The centers \(b_k\) are considered as random variables distributed in such a way that the slits \(\Gamma_k = \{z \in \mathbb{C} : z = b_k + \frac{1}{2}l e^{i\alpha_k} \ (0 \leq t \leq 1)\}\) generate a set of uniformly distributed non-overlapping slits. Theoretically this distribution can be introduced as the distribution of the variable \(\mathbf{b} = (b_1, b_2, \ldots, b_N) \in Q^N\) with the restrictions \(|b_m - b_n| > \delta\) (we assume that \(\delta = \frac{1}{2}\), and \(\delta\) called the separation parameter) for \(m \neq n (m, n \neq 1, 2, \ldots, N)\). It should be noted that the slits \(\Gamma_k\) belong to cell \(Q\) in the torus topology when the opposite sides of \(Q\) are glued by pairs. The formal definition of the random variable \(\mathbf{b}\) has to be statistically realised for large \(N\) to get numerical results.

We introduce the function of density slits \(\rho(l, N) = N \left(\frac{l}{2}\right)^2\), where \(l\) is the length of the slit and \(N\) the number of slits per representative cell. We describe a constructive method to generate the distribution of slits in a representative cell. The algorithm of distribution generating is similar to random sequential adsorption (RSA) model (We consider the slits instead of individual disk). This algorithm generates a probability distribution \(U_\rho\) depending on the density. Next, using the Monte Carlo method, with a fixed parameters (number of objects in a cell and the number of experiments) we calculate the Eisenstein–Rayleigh lattice sums in some way to characterize the distribution of slits (see. [7]). Features of the distribution have practical applications, eg. in the case of modeling materials with cracks or study the behavior of swimming bacteria. Generation random distribution of slits can also be used to determine the effective properties of the composite with slits. For this purpose, we determine the inverse function to conformal mapping, described in [4]. In the next step, we use the method described in [2].
2.2. Basic sums

We follow [5] to introduce basic sums. Let \( b_k (k = 1, 2, \ldots, N) \) be a set of points. Let \( q \) be a positive integer; \( k_0 \) runs over 1 to \( N; m_j = 2, 3, \ldots \). Let \( C \) be the operator of complex conjugation. Introduce the following sum of multi–index \((m_1, \ldots, m_q)\)

\[
e_{m_1 \ldots m_q} := N^{-\frac{1}{2}} \sum_{k_0 \ldots k_q} E_{m_1}(b_{k_0} - b_{k_1}) \times E_{m_2}(b_{k_1} - b_{k_2}) \ldots \times E_{m_q}(b_{k_q} - b_{k_q}).
\]

Sums \( e_m \) becomes the Eisenstein–Rayleigh lattice sums \( S_m \) [5] in the case \( N = 1 \) since it is assumed for convenience that

\[
E_m(0) := S_m.
\]

The Eisenstein functions [10] are related to the Weierstrass function \( \wp(z) \) [1] by the identities

\[
E_2(z) = \wp(z) + S_2,
\]

\[
E_m(z) = \frac{(-1)^m \cdot 2^{m-2}}{(m-1)!} \wp(z), \quad m = 3, 4, \ldots
\]

Every function (4) is doubly periodic and has a pole of order \( m \) at \( z = 0 \). The sums (1) constitute the basic elements to calculate the effective conductivity [6] dependent only on the locations of inclusion.

2.3. Application to bacteria

We pay attention to experimental results partially presented in [8]. The images of \textit{bacteria subtilis} in 31 frames are used in computations. We use algorithms of image processing and analysis to determine number, centers, angles of inclinations and length of bacteria. The density of bacteria is calculated by the described formula. The results of the image processing and analysis are applied to computation of the values of the experimental \( e \)-sums. Comparing the experimental and theoretical distributions using basic sum, we conclude that the distributions of bacteria are not chaotic. The general results are presented in Table 1.

3. Random location of glued disks

We may also consider other models. For example, we can generate distributions permanently glued two disks (see Fig. 2). For this, we can use the algorithm of generating distributions which is similar to \textit{random sequential adsorption} (RSA) model (We consider the group of disks instead of individual disk). In this case, using a similar approach as in [2], we can determine the formula for the effective conductivity of the composite material with inclusions of this type. The result can be compared with the model on the conductivity of the composite material with inclusions of type ellipse.

4. Conclusions

The comparison of two sets of \( e_m \) sums (the chaotic theoretical and the real ones) demonstrate their difference that implies the evidence of collective behavior of bacteria (see Table 1).

Table 1: Comparison averaged basic sums for distributions of bacteria and theoretical of distributions (\( \rho = 0.25 \)).

<table>
<thead>
<tr>
<th>( \text{Re} {e_{22}} )</th>
<th>( \text{Re} {e_{33}} )</th>
<th>( \text{Re} {e_{44}} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>averaged basic sums for theoretical distributions</td>
<td>3.19655</td>
<td>28.6764</td>
</tr>
<tr>
<td>averaged basic sums for distributions of bacteria</td>
<td>3.14674</td>
<td>29.9525</td>
</tr>
</tbody>
</table>

Figure 2: Cell with inclusions (two disks glued).

References

The Free Material Design reduced to the Monge-Kantorovich problem

Slawomir Czarnecki*, Radosław Czubacki**, Tomasz Lewiński*, Paweł Wawruch**

1**Faculty of Civil Engineering, Warsaw University of Technology
Armii Ludowej 16, 00-637 Warszawa, Poland
e-mails: s.czarnecki@il.pw.edu.pl*, r.czubacki@il.pw.edu.pl**, t.lewinski@il.pw.edu.pl*, pawlo.alegoria@gmail.com**

Abstract

The problem of optimal forming of an elastic body of a given mass to make it as stiff as possible reduces to the Monge-Kantorovich equation. This result by Bouchitté and Buttazzo of 2001 turns applicable to the problem of optimum distribution of elastic moduli of an anisotropic body under the condition of the unit cost being measured by the trace of the Hooke tensor. The aim of the paper is to rearrange the Free Material Design (FMD) problem in its three versions (anisotropic, cubic and isotropic) to the Monge-Kantorovich problem to clear up its well-posedness and show the links with the Michell problem.

Keywords: free material design, non-homogeneous isotropic elasticity, compliance minimization, Monge-Kantorovich equation

1. Introduction

In the original setting of the Free Material Design (FMD) problem all the components $C_{ijkl}$ of the elastic moduli tensor are considered as unknown at each point of the design domain $\Omega$ to form the stiffest anisotropic structure among all the structures -transmitting a given load to a given supporting boundary - of given cost measured by the integral of the trace of $C$; $\text{tr}C$ being the sum of eigenvalues (or the Kelvin moduli) of the elasticity tensor, cf. Ref. [1]. The load is usually given at a part of the boundary. Let the linear form $f(.)$ represent the virtual work of the load; if the vector field $v=(v_i)$, $i=1,...,d$, (the dimension $d=2$ or $d=3$), represents a virtual displacement field (vanishing on $\bar{\Omega}_0 \subset \Omega$) then the value of the work of the load on this virtual field is represented by $f(v)$. For a given distribution of moduli $C_{ijkl}$ within $\Omega$ the applied and fixed load causes the displacement field $u$. The total stiffness of the structure is viewed as $1/f(u)$, where $f(u)$ is called the compliance, or the total compliance of the structure. The condition of maximization of the total stiffness under a given value $V$ of the cost of the structure (defined as the integral of $\text{tr}C$ over the design domain) determines the layout of elastic moduli $C_{ijkl}$ within each point of $\Omega$. The optimal anisotropy becomes ideally suited to the load applied, which leads to a singular result: only one Kelvin modulus of $C$ becomes non-zero; this property has already been discovered in Ref. [1].

In order to arrive at a non-singular material two approaches are possible. The first one considers many load variants. The extension of FMD is based on choosing a convex combination of compliances as a merit function, corresponding to subsequent load conditions. If the number of load conditions is greater or equal 6 ($d=3$) or greater or equal 3 ($d=2$), then the optimal Hooke tensor becomes non-singular, with all positive Kelvin moduli, cf. Refs. [5,6]. The second remedy is to subject the anisotropy to a given material symmetry. Imposing isotropy (the method is called then: Isotropic Material Design or IMD) optimal nonzero values of the bulk and shear moduli is obtained, even if a single load condition is considered. If the cubic symmetric tensor $C$ is assumed (the method is then called: Cubic Material Design, CMD), then one of Kelvin moduli vanishes and to make all three Kelvin moduli positive one should apply at least two load conditions.

In each version of the design method (FMD, CMD, IMD) the optimal compliance equals $\int (u^*)/V$, where $u^*$ stands for the displacement field of the optimal structure and $Z = \min \left\{ \int \|\tau\|_1 \mid \tau \text{ being statically admissible} \right\}$ (1) with $\tau = (\tau_{ij})$ being the tensorial measure representing virtual stress fields, $\|\tau\|$ being a certain norm of $d \times d$ matrices. The form of this norm depends on the kind of the material symmetry assumed.

The problem (1) is linked with the Monge-Kantorovich theory of transhipping problems. The formulation (1) appeared already in the paper by Bouchitté and Buttazzo [2] on the shape optimization problem of a materially homogeneous body with the total mass constraint.

The support of the measure which solves the problem (1) determines the shape of the optimal structure while the optimal elastic moduli are determined within the body. Thus the FMD problem encompasses and solves two problems simultaneously: of optimal forming the shape of the body as well as the optimal distribution of its elastic moduli.

The problem dual to (1) reads

$$Z = \max \left\{ f(v) \mid v \in \text{Lip}(\Omega,\bar{\Omega}_0), \|v\|_1 \leq 1 \ a.e. \ on \ \Omega \right\}$$ (2)

where

$$\|v\|_1 = \max_{\tau \in \mathbb{R}^d} \| \tau \|_{\mathbb{R}^d}$$ (3)

is the norm dual to that involved in (1); the space $\text{Lip}(\Omega,\bar{\Omega}_0)$ is a Banach space of Lipschitz functions of values in $\mathbb{R}^d$ and vanishing on $\bar{\Omega}_0$.

Remark

The problems (1), (2) are extensions of the formulations of the transhipping problem towards tensorial settings. If the field $v$ in (2) is a scalar field, then the bound in (2) assumes the form: $|v| \leq 1$. The test fields in (1) are vectors. In order to satisfy the differential conditions included in (2) it may by assumed: $\tau_1 = F_1$, $\tau_2 = -F_1^T$, which reduces the problem (1) to the so-
called least gradient problem. The properties of the solutions to the latter problem are known and several benchmarks are available. The level sets of the solutions are ruled surfaces. Extension of these results to the tensorial problem (1) is a challenge of the contemporary optimization of structural topology.

2. Specification of problems (1), (2) in design of anisotropy

Consider the case of a single load variant. In the original FMD formulation the following isoperimetric condition is imposed

$$\int \nabla \cdot \sigma \, dV = V$$  \hspace{1cm} (4)

on the unknown Hooke’s tensor \( \sigma \). The minimal compliance of the optimal structure equals \( Z^*/V \), where \( Z \) is given by (1) or (2). In the considered version of FMD the norm in (1) is the Euclidean norm: \( \| \| = \| t \| = (\tau_\alpha \tau_\alpha)^{1/2} \). The norm (3) dual to the latter norm is also expressed by \( \| \| \). Thus the principal virtual strains lie within the unit ball.

Recently Czarnecki [3] proposed an isotropic version of FMD, called IMD. Unknown are distributions of the bulk \( k \) and shear \( \mu \) moduli within an isotropic body of minimal compliance. In case of \( d=3 \) the representation of the Hooke tensor reads

$$C = 3J + 2\mu \Lambda,$$  \hspace{1cm} (5)

$$J = \frac{1}{3} I \otimes I, \quad A = I - J, \quad I = \frac{1}{2} (\delta_\alpha \delta_\beta + \delta_\beta \delta_\alpha),$$  \hspace{1cm} (6)

the eigenvalues of tensor \( C \) being \((3k, 2\mu, 2\mu, 2\mu, 2\mu)\). Thus \( \text{tr} C = 3k + 10\mu \). One can prove that problem (1) in IMD involves an integrand expressed by

$$\| \tau \| = \alpha \| \tau \| + \beta \| \text{dev} \tau \|$$  \hspace{1cm} (7)

where \( \text{dev} \tau = \tau - (1/d) \text{tr} \tau \), \( \alpha = \sqrt{10}, \quad \beta = 5\sqrt{6} \), for \( d=3 \).

In paper by Czarnecki et al. [4] a cubic version CMD of the FMD method is proposed, i.e. the structure designed is to be made from a material of cubic symmetry at each point. This symmetry applies to the spatial case of \( d=3 \). A cubic representation of tensor \( C \) is expressed by Walpole’s formula [8]

$$C = aJ + bL + cM,$$  \hspace{1cm} (8)

where \( L = I - S, \quad M = S - J \) and

$$S = m \otimes m \otimes m \otimes m \otimes n \otimes n \otimes n \otimes n + p \otimes p \otimes p \otimes p \otimes p.$$  \hspace{1cm} (9)

In the CMD problem (or in the cubic material design) the design variables are the non-negative moduli \( a, b, c \) and the mutually orthogonal unit vectors \( m, n, p \). The eigenvalues of tensor \( C \) are \((a, b, b, c, c)\), hence \( \text{tr} C = a(3b+2c) \). The minimum compliance problem with the isoperimetric condition (4) comes down to the problem (1) with the integrand (7) in which, actually \( \alpha = \sqrt{3/3} \), \( \beta = \sqrt{2} \). The reader is referred to the examples presented in Czarnecki et al. [4]. Numerical results were found by solving subsequent variants of problem (1) with appropriate integrands. The solutions were constructed by the method proposed in Ref. [7], the minimizer of (1) determines the optimal components of the tensor \( C^* \) directly. The problem (2) is theoretically easier to handle, yet the maximizer \( v^* \) does not determine the tensor \( C^* \), but rather some selected characteristics of the solution.

3. Concluding remarks

Three versions of optimum design of anisotropy for a single load case were constructed: FMD, CMD and IMD. The main results concern the structure of the optimal Hooke tensor and of the layout of the elastic moduli. The optimal Hooke tensors are characterized by:

a) FMD: modulus \( \lambda^* \) of multiplicity 1 and five zero moduli,

b) CMD: optimal Kelvin modulus \( a^* \) of multiplicity 1, the modulus \( b^* = 0 \) of multiplicity 3, modulus \( c^* \) of multiplicity 2,

c) IMD: the Kelvin modulus \( k^* \) of multiplicity 1 and modulus \( \mu^* \) of multiplicity 5.

Thus only the IMD designs are non-singular in general. But even the isotropic design can be degenerated in some regions where one of the moduli vanishes. The condition of positive semi-definiteness of the isotropic tensor \( C \) in the IMD approach implies the bounds on the optimal Poisson ratio: \(-1 < \nu^* < 1/2 \) (\( d=3 \)) and \(-1 < \nu^* < 1 \) (\( d=2 \)). The optimal designs make use of the whole range of possible variation of the Poisson’s ratio; in particular, the negative values of this ratio occur in some sub-domains. Thus the auxetic materials appear as a result of compliance minimization.

References


Numerical aspects of patient specific material calibration of human artery: case study using clinical data

Tomasz Gajewski1, Hubert Stępak2, Krzysztof Szajek3, Tomasz Łodygowski4, Michał G. Stanisić5, Grzegorz Oszkinis6
1,3,4 Institute of Structural Engineering, Poznan University of Technology
Piotrowo 5, 60-965 Poznan, Poland
e-mail: tomasz.gajewski@put.poznan.pl
1,3,4
2,5,6 Department of General and Vascular Surgery, Poznan University of Medical Sciences
Długa 1/2, 61-848 Poznan, Poland

Abstract

This paper presents calibration of mechanical parameters of artery within inverse study problem fed by patient-specific data recorded before, during and after balloon angioplasty. Recorded data capture undeformed and deformed artery state and refers to the following medical imaging: three-dimensional magnetic resonance, two-dimensional angiography and inflation device pressure profile applied by a medic during angioplasty. After intervention finite element method model with constitutive parameters to characterize is built in order to mimic the clinical procedure. Iterative minimization procedure allows to identify crucial unknown mechanical constants of patient specific artery. A numerical model employs residual stresses inside layered artery wall and physiological state of soft tissue. The paper points out assumptions, limitations and crucial aspects of inverse analysis (IA) framework.

Keywords: inverse problem, soft tissues, biomechanics, medical imaging, patient-specific

1. Introduction

Balloon angioplasty is one of the common and established endo-vascular procedure which is routinely performed for patients with atherosclerosis. Therefore for scientific purpose could be benefited to extract mechanical parameters of artery from patient examination. For this purpose IA is performed.

Inverse analysis was employed by scientists and engineers for material calibration from many decades. Direct problem, or sometimes called forward problem, determines effects from causes, where the inverse problem in the opposite estimates causes from effects.

This paper deals with specific medical/clinical inverse problem. The aim is to build numerical framework of soft tissue mechanical identification, performed by using real patient balloon angioplasty. In addition, some crucial aspects of the setup will be pointed out. The numerical framework of backward analyses includes several essential components: the experimental/measurable quantities recorded during test, the numerical quantities obtained from a mathematical model able to mimic experimentally observed phenomena, an efficient minimization algorithm for improving convergence to the experimental results of a predicted field and an initial guess of investigated parameters. The unknown input parameters of system can be estimated by decreasing thorough iterations the discrepancy between two fields (measured and computed from the model). The framework applies medical imaging data as the measured field, namely pre and post magnetic resonance imaging (MRI) measurements and during intervention angiography recordings.

2. Finite element modelling

2.1. Geometry and mesh

After a patient consent, individual-specific geometry is captured from patient subjected to balloon angioplasty at Clinical Hospital of General and Vascular Surgery of Poznan University of Medical Sciences. The acquisition procedure was approved by the Bioethical Committee of the Poznan University of Medical Sciences.

From single intervention three geometries will be employed, (1) three-dimensional MRI gained no longer than 12 hours before the intervention, (2) three-dimensional MRI obtained no longer than 12 hours after the intervention and (3) two-dimensional angiographic data recorded during the intervention.

Medical imaging data is subjected to segmentation processing in order to prepare finite geometry, which later will be meshed for finite element method (FEM) engagement. The mesh preparation example is presented in Fig. 1, where preliminary tests to obtain solid artery mesh are presented.

Figure 1: Example of three-dimensional medical data segmented to finite surface geometry and processed to acquire finite element solid mesh.

*This research was supported in part by the Young Staff grant at the Poznan University of Technology (01/11/DSMK/0505).
2.2. Soft tissues cruxes

Soft tissue state of the art knowledge obliges to introduce residual stresses, physiological state of the structure and adequate medical device load conditions.

Residual stresses contribution are ensured by introducing prestretch ratios, according to Ref. [1], on element Gauss point level and submitting relaxation simulation in order to obtain deformation gradient input for regular analysis.

A physiological state of artery is ensured by preliminary simulation with simplified vascular geometry. In inverse trial and error analysis, blood pressures from patient measurements are applied to cylindrical artery. The subtask investigation aims to obtain initial zero-pressure system, for which computed and medical case deformed radii will match. Except physiological pressure other crucial load condition should be considered.

During a balloon angioplasty polymeric balloon is expanded inside lesion in order to increase vessel diameter. In the study balloon is inserted into the artery with material data specific for clinical device and expanded according to inflation pressure profile recorded during the intervention.

2.3. Constitutive modelling of artery

Selecting adequate physical law for describing mechanical behaviour of the artery is one of the most important task in the presented research.

Complex model engagement, with numerous parameters, will lead to multi-parameter inverse (optimization) problem, where a non-unique solution is more probable and calculations are more expensive in the meaning of computation time. On the other hand, the model result range of a simple constitutive law could be insufficient to properly mimic mechanical behaviour of the artery. As the first material model extended Gasser-Ogden-Holzapfel (GOH) model, from Ref. [2,3], was tested, however too large number of parameters excludes its full form from time efficient simulations, therefore some parameters could be fixed and excluded from IA with values based on the literature reports.

Material homogenization techniques can be taken into consideration as a reliable alternative to complex material models. This approach requires some additional implementation effort, however with later benefits of shorter inverse computations and more accurate results, than in a simple model.

2.4. Constitutive model of balloon

Angioplasty balloons are usually made from nylon 12, silicone, polyethylene terephthalate or polyurethane. Depending on the balloon applied during the intervention, particular material will be modelled in finite element code. Mechanical behaviour of balloons used during angioplasty are usually characterized as isotropic and hyperelastic, therefore neo-Hookean or second order Ogden model will be adapted.

3. Inverse problem framework

The inverse problem takes the least square form, where the following function is minimized:

\[ f(x) = \sum_{i} (U_{\text{EXP}}^i - U_{\text{NUM}}^i(x))^2, \]  

where \( f(x) \) is the objective function with unknown set of design (constitutive) parameters, \( x \). \( U_{\text{EXP}} \) and \( U_{\text{NUM}} \) are the measurable/computed value, where \( i \) is changing from 1... to \( n \) and stands for the number of field value.

Displacements of inner or/and outer surface of artery at balloon inflation area are employed as the calibration data, mentioned as the measurable/computed quantities.

Within an inverse problem in the case of too small amount of measured data, an ill-posed problem which can lead to non-unique solution. Here, in order to avoid ill-posed problem may be obtained, not only before and after artery geometry is utilized (MRI), but also during intervention (angiography). This data enrichment enlarges robustness of correct calibration outcome.

The GOH constitutive law employment in three layer artery (intima, media and adventitia) model requires over a dozen parameters to define, therefore some of minor importance will be excluded from IA. Model unknowns selected for IA calibration are \( k_i \) – fiber families dispersion and \( \mu_i \) – shear modulus, where \( i \) is for intima, media and adventitia. Therefore a greater number of material parameters is reduced to six.

4. Summary

Despite years of investigating by scientists/engineers, patient material characterization of soft tissues is still very challenging task. Impossibility of direct in-situ and ex-vivo typical mechanical tests on living tissues, such as biaxial testing, enforces mixed numerical-clinical approaches with uncertainty of geometrical measurements, necessity of some data assumptions and simulations simplifications. Those aspects should be proceeded with exceptional care in order to obtain valuable material parameters.

The authors run-through-problem allows to list some crucial aspects which should be developed in the future inverse problem framework in the case of successful material characterization coupled with clinical intervention for a living human individual. Although, the regular inverse analysis will be studied in the nearest future, at the present stage of investigation the following main points could be listed:

- model with material homogenization could be more time efficient then complex model, and more accurate than simple approach,
- for multiple parameters artery model, i.e. GOH model, minor parameters should be fixed to obtain time efficient IA,
- except before and after intervention data, also during intervention information should be used to avoid ill-posed problem.

References


Temperature dependencies of ultrasound signals backscattered from an agar-oil soft-tissue mimicking material

Barbara Gambin\textsuperscript{1}, Eleonora Kruglenko\textsuperscript{2}, Wojciech Secomski\textsuperscript{3}, Piotr Karwat\textsuperscript{4}

\textsuperscript{1,2,3,4}Department of Ultrasound, Institute of Fundamental Technological Research of the Polish Academy of Sciences Pawińskiego 5B, 02-106 Warszawa, Poland
e-mail: bgambin@ippt.pan.pl \textsuperscript{1}, ekrug@ippt.pan.pl \textsuperscript{2}

Abstract

Tissue mimicking materials for ultrasound research, phantoms, should be acoustically similar to the tissues. Such requirements are filled by the AGO (agar-oil) phantoms. Here, they were used in the experiment of heating internal region of samples by a high intensity ultrasound (HIFU) transducer. During heating the RF (radio frequency) ultrasound signals were collected. It is demonstrated that the temperature changes in AGO phantoms can be described by special properties of the backscattered RF signals, namely the shape parameter of the Nakagami distribution and SNR (signal to noise ratio) of signal envelope random distribution. The revealing of qualitative relationships between the temperature increase/decrease measured by thermocouples and the statistical parameters changes are main results of the paper.

Keywords: soft tissue phantom, absorption of acoustic energy, temperature marker, signal-to-noise ratio, Nakagami distribution

1. Introduction

Phantoms intended for quality assurance and performance assessment of imaging systems must have acoustic properties similar to those exhibited by a soft tissue. The well-characterized phantoms are required to validate procedures that attempt to generate quantitative ultrasound feature images based on these properties. The statistical properties of a received signal envelope are now often used as one of the methods for the differentiation of soft tissues \textit{in vivo} and \textit{in vitro}, cf. [6]. The thermal characterization of one type of AGO by the statistical characteristics of backscattered signals has already been done in [4]. In the paper four different types of samples are used. They are characterized by different oil content, what follows different acoustic energy absorption. Section 2 contains a description of phantoms, experiment procedures and methods. In Section 3 the results and final remarks are given.

2. Materials and Methods

2.1. Agar-oil phantom

Different types of samples used in experiments were prepared as a mixture of water agar solution and from edible safflower oils, see Table 1.

<table>
<thead>
<tr>
<th>Phantom name</th>
<th>Oil in %</th>
<th>Water in %</th>
<th>Agar in %</th>
</tr>
</thead>
<tbody>
<tr>
<td>AGO 10</td>
<td>10</td>
<td>89.10</td>
<td>0.90</td>
</tr>
<tr>
<td>AGO 25</td>
<td>25</td>
<td>74.25</td>
<td>0.75</td>
</tr>
<tr>
<td>AGO 33</td>
<td>33</td>
<td>66.33</td>
<td>0.67</td>
</tr>
<tr>
<td>AGO 50</td>
<td>50</td>
<td>49.50</td>
<td>0.50</td>
</tr>
</tbody>
</table>

Small amount of surface tension reducing agent, dishwashing liquid was added to obtain more fine emulsion. Production of such phantom, called AGO (agar-oil) has been documented in our previous paper [4]. In the image of a phantom microstructure the mixing process is visible in characteristic texture formed from the oil phase in water- agar solution, cf. Fig. 1.

![Figure 1: AGO material image made with optical microscope (Nikon Microphot) with the lens Nikon BD Plon 5x0.1](image)

2.2. Experiment description

The heating of the samples was performed using a system consisting of a generator (Agilent 332, Aprings Colorado, USA), an amplifier (ENI 1325LA, Rochester NY, USA), a spherical ultrasonic transducer (central frequency 2.2 MHz, diameter 44.5 mm, 44.5 mm focal length, area $S = 15.2 \text{ cm}^2$) and an oscilloscope (Tektronix TDS3012B). Irradiation of the samples with a series of acoustic bursts (20 cycles of 2 MHz sine wave repeated every 50 micros) of average acoustic power equal 6 W was performed. During 10 minutes of heating and 10 minutes of cooling the temperature changes were measured using thermocouples and registered by a USB module - TEMP. The temperature within the sample was measured along the beam axis at distances of 30, 35, 40, 45, 50 mm from the transducer. The geometrical focus was located 44.5 mm from the surface of the transducer, while the maximum temperature was observed at a distance of 40 mm. An ultrasound imaging system (Sonix TOUCH, ULTRASONIX, British Columbia, Canada) equipped with a 128-elements linear probe (L14-5/38) was used for acquisition of the ultrasonic radio frequency (RF) echoes from the heated sample. The probe was located transversely to the axis of the heating beam at a distance of 40 mm from the heating transducer. The sample was scanned every 5 sec during 20 min of the experiment. Each scan was performed in a 200 ms break in the heating to ensure that

\footnotetext{This work was partially supported by the National Science Centre (grant no. 2011/03/B/ST7/03347).}
imaging process is free from interference coming from the heating beam. The probe elements were driven by 2 periods of 6.7 MHz sine and the scanning was done with use of the synthetic aperture technique. In a single transmit-receive cycle a pair of adjacent elements emitted a spherical wave and all supported elements received the echoes. In successive transmit-receive cycles the transmit aperture was moved by a single element. In our experiments only 42 probe elements were used due to the limitation of the transmit sequence length in the Sonix TOUCH system. The collected RF echoes were reconstructed off-line with use of the Matlab software.

2.3. Methods

We use the simplified Pennes model bio-heat equation without perfusion for modeling the heat transfer in agar oil (AGO) soft tissue phantoms with special source produced by HIFU (high intensity focused ultrasound) transducer, cf. [2]. The result of FEM calculation allowed to determine the region of samples which were heated, cf. Fig. 3. Next, the signals from this regions were used to perform the statistical analysis. At first, the attenuation compounding is performed same as [1]. Than envelopes were obtained by the Hilbert transform. The optimization of the choice of a parameter to measure temperature variation has been proceeded similarly as in [4]. The optimality criteria used consist the quality of matching histograms to probability distributions and the highest sensitivity to temperature changes. The chosen parameters are the shape parameter of the Nakagami distribution and SNR (signal to noise ratio). They were calculated for four types of AGO materials in 1200 time points during the heating process, see Fig. 4.

![Figure 2: FEM simulation of temperature distribution and the heated region of sample](image)

3. Results and conclusions

The shape parameter of the Nakagami distribution and the SNR variations during the heating process are shown in Fig. 3. There are much greater fluctuations in parameters values during heating than cooling. This may be caused by a long relaxation times (viscoelastic effect) or by the random interference with signals from the heating transducer. The second observation is that the changes of AGO 10 and AGO 50 parameters exhibit worse effect of the temperature rise, cf. in Fig. 3.

![Figure 3: SNR and Nakagami parameter for AGO 25 and AGO 33 as functions of time](image)

The temperature measured by thermocouples as a function of time was fitted to an exponential function with the high values of goodness of fit measures. Similarly, the fittings are done for the Nakagami parameter as a function of time, see Fig. 4.

![Figure 4: Comparison of Nakagami parameter changes and measured temperature changes during heating/cooling process](image)

The rise and fall of the Nakagami parameter values is consistent with the changes in the temperature measured by thermocouples, besides the rates of changes reflect the differences between heating and cooling. Independently, the FEM modelling of temperature fields can be used to approximate the thermal conductivities of different AGO materials. This technique was recently presented in [5], when the thermal conductivity of the soft tissue sample in vitro was obtained.

References


Changes in ultrasound echoes of a breast tissue \textit{in vivo} after exposure to heat - a case study.

Barbara Gambin\textsuperscript{1}, Eleonora Kruglenko\textsuperscript{2}, Michał Byra\textsuperscript{1}, Andrzej Nowicki\textsuperscript{1}, Hanna Piotrzkowska-Wróblewska\textsuperscript{2}, Katarzyna Dobruch-Sobczak\textsuperscript{6}

1, 2, 3, 4, 5, 6 Department of Ultrasound, Institute of Fundamental Technological Research of the Polish Academy of Sciences Pawiiakiego 5B, 02-106 Warszawa, Poland, e-mail: bgambin@ippt.pan.pl

6 Cancer Center and Institute of Oncology, Maria Skłodowska-Curie Memorial Wawelska 15, 02-034 Warsaw, Poland

Abstract

A B-mode ultrasonography provides structural information on the tissue under investigation encoding the echo strength in gray scale in a two-dimensional image. Interpretation of the B-mode image of breast tissue is done by a physician. The analysis of statistical properties of backscattered RF signal has been recently applied successfully to distinct healthy tissue from tissue lesions regions as a new method of quantitative ultrasound (QUS). Up till now, the most reliable results were obtained for liver and renal tissue lesions, because their normal, healthy structures are nearly homogeneous while a heterogeneous breast tissue classification is still an open issue. The recent study revealed that the medium contraction and expansion induced by a temperature change can cause variations in the relative position of scatterers in a tissue. We have developed a new procedure of heating the patient breast and allowing to observe and record \textit{in vivo} the influence of temperature changes on a B-mode image and properties of unprocessed radio frequency (RF) backscattered echoes. The initial, feasibility studies of influence of the temperature increase in breast tissue on the intensity, spectrum and statistics of ultrasonic echoes will be discussed.

Keywords: breast tissue, RF signal, backscattered signal amplitude statistics, spectral properties

1. Introduction

A B-mode ultrasonography provides structural information on the tissue under investigation encoding the echoes strength in gray scale in a two-dimensional image. Interpretation of the B-mode image of breast tissue is done by a physician. Recently new methods of quantitative ultrasound (QUS) dedicated to estimation of structural changes in tissue are being developed, cf. [4], [6]. Particularly, analysis of statistical properties of backscattered RF signal has been successfully applied to distinct healthy tissue from tissue lesions regions, see [5]. Up till now, the most reliable results were obtained for liver and renal tissues lesions, because their normal, healthy structures are nearly homogeneous while a heterogeneous breast tissue classification is still an open issue. A recent study revealed that the medium contraction and expansion induced by a temperature change can cause variations in the relative position of scatterers in a tissue. We have developed a novel algorithm of heating the patient breast and recording the changes in ultrasonic B-scans and in RF echoes. In the statistics of the signal information is encoded about the type of dissipative structures. The structural information on the type of dissipative tissue is encoded in the statistics of the returning signals. When a signal is backscattered from a large number of uniformly distributed scatterers then random amplitude fluctuations are close to the Rayleigh distribution. Variations of the structure to a more heterogeneous in terms of the appearance of cell clusters in the distribution of scatterers or the scatterers reflectivity variations result in the amplitude distribution close to K-distribution or Nakagami distribution. The experiments were carried out in order to evaluate the temperature influence on variation in statistical distributions of signals recorded in tissue phantoms and samples of soft tissue \textit{in vitro}, cf. [1], [2] and [3]. In what follows the initial, feasibility studies of influence of the temperature increase in breast tissue on the signal intensity, spectrum and statistics of ultrasonic echoes will be discussed. The paper is organized as follows: Sections II introduce the experiment description and methods, Section III presents the results. In Section IV conclusions are drawn about the contributions of this study to the general ultrasonography.

2. Material and Methods

2.1. Experiment description

Backscattered ultrasound RF signals and B-mode images have been collected using ULTRASONIX, (SonixTOUCH, Canada) scanner, and standard linear array L14-5/38, 10 MHz transducer, focused at the depth 3,5 cm. The patient breast was scanned before and after heating by physician who has localized the lesion region to be analyzed. The heating process was done directly through the skin to which a rubber seal bag with hot water at c 50°C during 10 minutes was applied.

2.2. Methods

The signal frequency spectrum from the regions at two temperature levels has been also obtained. Next, for each sub regions of tissue the statistical parameters have been correlated to scatterer strength, density and distribution. All scan lines were compensated for attenuation and then the amplitudes of all FR lines have been calculated using of Hilbert transform. Next, the fitting of amplitude histograms to different probability density functions together with the evaluation of the accuracy of matching by MSE (mean square error) was performed.

\textsuperscript{*}This work was partially supported by the National Science Centre (grant no. 2011/03/B/ST7/03347).
3. Results

A B-mode image was recorded from the same part of breast before and after heating. Zoomed region of the heated breast regions are presented in Fig. 1.

![B-mode images before and after heating](image)

Figure 1: B-mode of breast tissue before and after heating.

The thermal behaviour of spectral properties of backscattered signal are shown in Fig. 2, in which the amplitude and the bandwidth changes of the spectrum are marked.

![Mean spectrum in cyst and tissue regions before and after heating](image)

Figure 2: Mean spectrum in cyst and tissue regions before and after heating.

The statistical properties of signal amplitude are described by two parameters. The SNR (signal to noise ratio), here the ratio between mean value and standard deviation, and the shape or scale parameter of best matched to the amplitude histogram distribution are calculated similarly as in [3]. In Figure 3 the matchings are shown and the chosen characteristic parameter value are given before and after heating of tissue and cyst regions, respectively.

![Histogram of signals amplitudes collected from tissue region marked on B-mode and fittings to different probability density functions](image)

Figure 3: Histogram of signals amplitudes collected from tissue region marked on B-mode and fittings to different probability density functions.

In order to illustrate the statistical parameters relation between SNR shape parameter of K-distribution is shown in Fig. 4.

![Plot SNR versus the shape parameter of K-distribution for different regions of breast](image)

Figure 4: Plot SNR versus the shape parameter of K-distribution for different regions of breast.

4. Conclusions

Note that the B-mode image quality is significantly improved after heating. The fine details of the cyst structure are clear while the same region before heating was simply dark, water-like. The depth of penetration of ultrasound was increased by about 0.5 cm comparing to the pre-heating scanning, cf. Fig. 1. In the less heated regions (left of the cyst) no visible differences occur, however they are clear when calculated versus temperature change, cf. Figure 6. It should be stressed that the sensitivity of statistical parameters to temperature is very high, particularly in the cyst region as compared to the normal tissue regions. This effect can be explained by a larger increase in speed of sound in a fluid-like cyst volume, than in an inhomogeneous tissue region. After heating the spectrum amplitude increases much larger in the lesion than in the surrounding tissue. In a liquid medium the rise of a spectrum level is much greater than in tissue region.

References


Digitally reconstructed radiograph procedure for modifying 3D model to meet the intraoperative vertebrae location

Dominik Gaweł1*, Paweł Główka1, Michał Nowak2

1,2Department of Pediatric Orthopaedics and Traumatology, Poznan University of Medical Sciences
28 Czerwca 1956 135/147 street, 61-345 Poznań, Poland
e-mail: dominik.r.gawel@doctorate.put.poznan.pl

Abstract

The paper describes an effective method of creating digitally reconstructed radiograph images and applying them for a 3D intraoperative visualization. Developed algorithm is based on standard mathematical calculations reducing the necessity to use sophisticated software and libraries. The algorithm requires no special training to use it. The presented solution allows to create digitally reconstructed radiographs (DRR) from computer tomography (CT) and magnetic resonance (MRI) data on every desktop computer even by inexperienced users. The system requirements of the algorithm are incomparably lower in comparison to other common DRR techniques. The presented algorithm was developed to allow the modifying 3D spine model, created from CT images, to meet the intraoperative vertebrae location. Such solution gives an accurate 3D visualization of patient internal structures during the surgery.

Keywords: digitally reconstructed radiograph, computed tomography, magnetic resonance imaging

1. Introduction

Medical imaging is a branch of diagnostics used to create images of the human body for diagnostic and treatment purposes. Over the last few dozen years a great development in medical imaging took place. New technologies were discovered, allowing for a more accurate and complex diagnostics [1,2].

The first developed medical imaging technique was radiography. It uses electromagnetic, phenomena especially x-ray radiation, to view the internal structure of a human body, providing its 2D representation.

Another medical imaging technique using x-ray radiation is computed tomography. The CT produces volume images, providing a 3D representation of internal structures.

Magnetic resonance is slightly new, non-harmful, medical imaging technique. In this technique strong magnetic fields and radio waves are used to form a three-dimensional representation of internal human body structures.

2. Pre-operative procedures

In the domain of evolutionary spine deformations, due to ductility of a bone tissue subjected to asymmetrical loads, scoliosis occurs. Both spatial layout of the vertebrae and their shape are incorrect. As a preparation to the spinal implant surgery, a pre-operative spine evaluation is made. Standard pre-operative procedures consist of creating an individual da Vinci presentation for every patient [3]. It is a representation of the spine seen from the cephalad side in a horizontal plane. Such solution is complementary to the standard projection of the spine in coronal and sagittal planes.

The most important part of pre-operative measurements is finding the central points of vertebrae. Currently this procedure is made on the basis of x-ray images that are taken in a standing position, while surgery is made in a lying position. The use of CT/MRI images eliminates this problem, but greatly increases the time required for proper preparation to the surgery. The presented solution converts CT/MRI data into single coronal and sagittal two-dimensional images similar to the standard computed radiographs [4].

3. DRR procedure

At the beginning DICOM (Digital Imaging and Communications in Medicine) images are read and converted into PNG file format.

Next, the most important part of the algorithm begins. Firstly three-dimensional array containing values of every pixel from every single image is created. On this basis a mean value of every column and row in X, Y and Z directions is calculated. The results are stored in three two dimensional arrays presenting base medium images in Axial, Coronal and Sagittal directions. Those base images are used for further calculations.

On the basis of the user-specified values and medium pixel values, for each row and column significance boundaries are calculated. Pixel values located inside the boundaries are recognized significant to create final digitally reconstructed radiograph image. Pixels with values located outside the boundaries are recognized as background, not used in further calculations.

The next stage consists of creating final image for each of elementary plains. Mean values of the significant pixel values for each column and row are calculated and stored in the final array. Afterwards two dimensional images in PNG file format are created. Each pixel value in the final array is assigned to the corresponding pixel in the final image, creating a digitally reconstructed radiograph. On the basis of data spacing and slice thickness values from the DICOMDIR file, a global coordinate system is calculated [5]. The final image is scaled to fit the specified parameters and export to DICOM file format, allowing for further measurements.

*This work was supported by The National Centre for Research and Development under the grant - decision no. DZP/PBS3/2296/2014.
4. Results

Due to cooperation with multiple medical facilities a wide variety of medical imaging data was obtained for tests. All supplied image sets were in DICOM file format which is a standard medical information exchange format.

As an example of CT data a spine presenting adolescent idiopathic scoliosis was chosen. The data was created as a part of pre-operative evaluation facilitating measurements for spinal implants and supplied by Wiktor Dega Orthopedic and Rehabilitation Clinical Hospital.

As a result of the presented algorithm a digitally reconstructed radiograph from CT data was created (Fig. 1). Such reconstruction allows to perform the pre-operative measurements much easier and faster than in the case of standard image sets. Creating the DRRs instead of a standard computed radiography decreases costs, time and labour, eliminating the harmful effects of x-ray radiation [6].

The created DRRs can be used for 3D intraoperative visualization of patient vertebrae location. The Da Vinci presentation based on the DRR images defines position of vertebrae central points. The obtained data is combined with a 3D model created from the CT images and compared with an intraoperative radiograph. The 3D model is modified to meet the intraoperative vertebrae location (Fig. 2).

5. Conclusions

The work presents an effective method of creating digitally reconstructed radiograph images. The developed algorithm is based on standard mathematical calculations reducing the necessity to use of sophisticated software and libraries. There is also no need to go through any special training to use it. The presented solution allows to create digitally reconstructed radiographs from computed tomography and magnetic resonance image sets on every desktop computer even by inexperienced users. The system requirements for the algorithm are incomparably lower compared to other common DRR techniques.

References


Effective conductivity and critical properties of 2D composites

Simon Gluzman\textsuperscript{1}, Vladimir Mityushev\textsuperscript{2}, Wojciech Nawalaniec\textsuperscript{3}

\textsuperscript{1}Bathurst 3000, Toronto M6B 3B4 Ontario, Canada
\textsuperscript{2,3}Department of Computer Sciences and Computer Methods, Pedagogical University,
ul. Podchorazych 2, Krakow 30-084, Poland

e-mails: mityu@up.krakow.pl \textsuperscript{2}, wnawalaniec@gmail.com \textsuperscript{3}

Abstract

Two-phase composites with non-overlapping inclusions embedded in a matrix are theoretically investigated. The effective conductivity is expressed in the form of a series in the volume fraction of ideally conducting disks. The problem of direct reconstruction of the critical index for superconductivity from the series is solved with good accuracy, for the first time. General analytical expressions for conductivity in the whole range of concentrations are derived for regular and random composites, compared with the existing models.

Keywords: regular composite, random composite, effective conductivity, critical index, crossover formula

1. Introduction

The 2D, two-component composite made of a collection of non-overlapping, identical, ideally conducting circular disks are embedded randomly in an otherwise uniform locally isotropic host (see Fig. 1). The effective conductivity problems for an insulating or ideally conducting inclusions are called the conductivity and superconductivity problems, respectively. The problem and its approximate solution goes back to Maxwell.

Two important unresolved problems exist in the theory of random composites: 1. the quantity to stand for the maximum volume fraction \( x_c \) of random composites and 2. theoretical explanation of the values of critical indices for conductivity and superconductivity denoted by \( t \) and \( s \), respectively.

Recently, a novel technique for deriving expansions in concentration was suggested [5, 4]. It combines analytic and numeric methods for solving the conductivity problem directly in the 2D case. It is applicable both for regular and random cases [2, 3, 8]. Thus, we proceed to the case of 2D composites, where the series in concentration for the effective conductivity by itself, will be presented and analyzed systematically, following generally to [2, 3, 8] and partially presented in [4]. The series will be used to estimate the index and the threshold in 2D random case.

The problem of defining the threshold is highly non-trivial, since the random closest packing of hard spheres turned out to be ill-defined, cannot stand for the maximum volume fraction. It depends on the protocol employed to produce the random packing as well as other system characteristics.

The problem seems less acute in two dimensions, where various protocols agree on the quantity to stand for the maximum volume fraction of random composites. It is the concentration of \( \frac{\pi}{\sqrt{12}} \approx 0.9069 \), attained only for a regular hexagonal array of disks. The sought value was long estimated to 0.82, and considered a random close packing value. It was recognized recently, that it does not correspond to the maximally random jammed state [4]. For the volume fractions above 0.82 a local order is present and irregular packing is polycrystalline, forming rather large triangular coordination domains-grains. In this paper a protocol leading to \( x_c = \frac{\pi}{\sqrt{12}} \) is used, although our method can be applied with another protocol of an unknown \( x_c \).

All attempts to explain the value of critical indices through geometrical quantities of a percolation problem, i.e. universally, have failed so far, the indices are considered independent. From the phase interchange Keller’s theorem it follows that in two-dimensions the superconductivity index is equal to the conductivity index.

While it is clear that using expansions in concentration for the conductivity, it is possible to address the two problems, in practice there are no more than two terms available for random systems, because of serious technical difficulties. No method even such powerful as renormalization, or resummation approaches can draw reliable conclusions systematically, based on such short series [1]. “In fact, the age-old method of series expansions is also blocked by the same difficulties...”[1].

This concerns also the whole family of self consistent methods (SCMs) which include Maxwell’s approach, effective medium approximations, differential schemes etc. SCMs are valid only for a dilute composites when interactions between inclusions do not matter [7]. The idea to correct a self consistent method (SCM) result \( t = s = 1 \) (in all dimensions) using the series in concentration remained, therefore, practically unattainable.

We should also mention an indirect approach to estimating \( t \) for resistor networks from resistive susceptibility by scaling rela-

Figure 1: Randomly distributed disks.
2. Series for conductivity

In order to correctly define the effective conductivity tensor \( \sigma \) of random composites, the random distribution of disks of radius \( r \) must be introduced, since the second order term of \( \sigma \) in concentration depends on the distribution \([7]\). For macroscopically isotropic composites, the third order term begins to depend critically even when a concentration depends on the distribution \([7]\). For macroscopically isotropic composites, the third order term begins to depend critically even when a

\[ \sigma(x) = 1 + 2x + 2x^2 + 5.00392x^3 + 6.3495x^4 + O(x^5). \]  

The coefficients on \( x^k \) \((k = 5, 6, 7, 8)\) vanish in \((2)\) with the precision \(10^{-10}\). Following the method \([4]\), we obtain reconstruct the effective conductivity explicitly

\[ \sigma^*(x) = \left( \frac{x(0.65 - 1.17x) - 0.04}{x(0.65) + 0.04} \right)^{-1/3}. \]  

For regular arrays of cylinders the index is much smaller, \( s = \frac{1}{4} \) \([4]\) and the critical amplitude is also known with a good precision. The critical behaviour of regular composites is not practically mentioned together with other critical phenomena. It is remarkable that a relatively "simple" Laplace equation for the potential, when complemented with a non-trivial boundary conditions in the regular domain of inclusions, behaves critically even without explicit non-linearity or randomness, typical to the phase transitions and percolation phenomena.

References

The impact of ergonomic factors influencing armoured vehicle crew safety

Marek Gzik¹, Wojciech Wolański², Bożena Gzik-Zroska³, Kamil Joszko⁴, Michal Burkacki⁵, Sławomir Suchoń⁶

¹,²,⁴,⁵,⁶ Faculty of Biomedical Engineering, Department of Biomechatronics, Silesian University of Technology
Roosevelta 40, 41-800 Zabrze, Poland
e-mail: marek.gzik@polsl.pl, wojciech.wolański@polsl.pl, kamil.joszko@polsl.pl, michal.burkacki@polsl.pl, slawomir.suchon@polsl.pl

³ Faculty of Biomedical Engineering, Department of Biomaterials and Medical Devices Engineering, Silesian University of Technology
Roosevelta 40, 41-800 Zabrze, Poland
e-mail: bozena.gzikzroska@polsl.pl

Abstract

All light armoured vehicles (LAV) mostly suffer from small crew compartment. Lack of space around the soldiers does not only affect their travelling comfort but also is one of the factors which affect their safety during IED or mine explosion. The aim of studies is to estimate influence of selected ergonomic factors on crew safety. Authors used Madymo software to build model of LAV with six soldiers in the crew section. Injury criteria were obtained based on NATO STANAG HFM-148 document which provided the guidelines concerning arrangement of seats in LAV. Hybrid III dummy was used with Mil-LX leg which is dedicated for blast purposes. Soldier equipment models were obtained during 3D scanning process. Different seat belt systems were analysed in case of not only local effects of explosion but also during rollover incident.

Keywords: ergonomic, soldier modelling, biomechanical analysis, shock wave, Madymo

1. Introduction

Ergonomic design of armoured vehicles is essential due to not only crew comfort and space usage but also safety. In this paper, authors are focused on infantry compartment ergonomics, set-up, seats allocation and general principles towards minimalizing risk of injury in case of IED or mine under-belly explosion.

Modelling was carried out in Madymo software, dedicated for automotive crash tests, also used to blast effect investigation. Significant advantage of numerical approach is an ability to separate influencing factors like input acceleration data, explosion scenario, occupants position etc.

2. Methods

2.1. Model assumptions

For injury investigation, numerical model of LAV and equipped soldier model was developed. According to NATO STANAG HFM-148 soldier’s model is based on verified Hybrid III model with modified Mil-LX leg. Polish combat helmet wz. 2005 type was add to soldiers dummy, which is important in case of head contact and due to inertia. Vehicle interior is equipped with current used seats with footrest and seat belts (Fig. 1).

In simulated scenario local and global effects of primary phase, rebound phase and resulting accident were considered. Authors assumed blast scenario without armour deformation, focusing only on resultant acceleration, hence pressure, temperature and ballistic factors were excluded. Input acceleration is taken from LS-DYNA experiment 0.

2.2. Injury criterion

To obtain injury risk occurred during rapid acceleration and deceleration of body parts injury risk criterion were assumed. This criterion set were introduced in HFM-148 and were determine basing on PMHS as well as blast experiments and were confronted with dummy response for full compatibility. Presented criterions are so far the best tool to determine risk of damaging human body in explosion threat [3].

For head there is HIC₃₆ criteria, widely used in other applications (including crash tests). For blast tests shorter 15 ms time period is used due to rapid phenomenon characteristic. HIC₁₅ is calculated basing on head acceleration (Eqn 1).

\[
\text{HIC} = \left[\frac{1}{(t₂ - t₁)} \int_{t₁}^{t₂} a(t)dt \right]^{1/3} (t₂ - t₁)
\]

where: \(a\) – resultant linear acceleration of head [m/s²]; \(t₂ - t₁\) – 15 ms time interval when HIC value is maximum [s]
Table 1: Belts systems comparison.

<table>
<thead>
<tr>
<th>Body part</th>
<th>Criterion</th>
<th>Tolerance level</th>
<th>Simulation result</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>two-points seat belts</td>
</tr>
<tr>
<td>Head</td>
<td>Head Injury Criterion (HIC15)</td>
<td>250</td>
<td>160÷1260</td>
</tr>
<tr>
<td>Lower limb</td>
<td>Axial compression force in tibia</td>
<td>2.6 kN</td>
<td>4.6kN</td>
</tr>
<tr>
<td>Chest</td>
<td>Thoracic Compression Criterion</td>
<td>30 mm</td>
<td>6 mm</td>
</tr>
<tr>
<td>Viscous Criterion</td>
<td>(VC frontal)</td>
<td>0.7 m/s</td>
<td>0.00419÷0.00482m/s</td>
</tr>
<tr>
<td></td>
<td>Axial compression force (-Fz)</td>
<td>4kN - 0 ms</td>
<td>6kN</td>
</tr>
<tr>
<td></td>
<td>1.1kN - 30ms</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Neck spine</td>
<td>Shear force (Fx+ / Fy+)</td>
<td>3.1kN - 0ms</td>
<td>3.2kN</td>
</tr>
<tr>
<td></td>
<td>1.5kN - 25-35ms</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1.1kN ≥45ms</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Lumbar spine</td>
<td>Bending moment (flexion) (My)</td>
<td>190 Nm</td>
<td>100Nm</td>
</tr>
<tr>
<td></td>
<td>Bending moment (extension) (-My)</td>
<td>96 Nm</td>
<td>140Nm</td>
</tr>
<tr>
<td></td>
<td>Dynamic Response Index (DRIz)</td>
<td>17.7</td>
<td>18.24</td>
</tr>
</tbody>
</table>

Second non directly measured criteria is DRIz, based on backbone spine model including weight subject to acceleration impulse by elastic and suppressing elements connected parallel. DRIz value is given by Eqn 2.

\[
DRI_z = \frac{\omega_n^2 \delta_{\text{max}}}{g}
\]  

where: \(\omega_n\) – constant, circular frequency, (52.9 [rad/s]); \(\delta_{\text{max}}\) – maximum compression calculated from motion equitation of spine model; \(g\) – constant, gravity acceleration (9.81 [m/s²])

3. Results

With the prepared model, several factors influencing safety were considered in tests: feet placement, knee joint angle, distance between passengers, surrounding space and different seat belts systems (Tab. 1).

Main work was focused on lower leg (Fig. 2) and head injury mechanism. Tibia compression force is over the tolerance level while placing feet on floor. In feet on footrest scenario there is significant difference between force occurring in left and right leg because of collision with other crew members and vehicle equipment. Head acceleration (Fig. 3) and HIC15 are dramatically different for two-point and five-point belt system. In first case head makes contact with roof which effects in high risk of severe injury (HIC15=1260); second case value is below limit level and there is no injury jeopardy (HIC15=180).

4. Summary

In this study numerical models influencing safety factors were estimated. The simulation results allowed to specify some principles enhancing armoured vehicle crew safety in case of IED or mine attack. The presented studies showed that there is strong influence of ergonomic set-up and other factors related to body position on injury risk. All tested scenarios are rather simple changes in vehicle equipment which are low cost efforts that can reduce the likelihood of injury.

Nevertheless, some future studies should be done for more precise investigation of this extensive problem. Another conclusion, is that similar studies should be done on vehicle development level. More space between roof and soldier head decrease impact possibility.

The Madymo software proves to be a useful tool in performed studies. Multibody, used in place of finite element method approach, allows to shorten numerical simulation time and also perform multitudinous scenario combinations.

Moreover, software provides easy factor isolation possibility which was necessary for carrying out the experiments.

![Figure 2: Tibia compression force feet on footrest and floor](image2)

![Figure 3: Head acceleration for different seat belts systems](image3)

References


Safety analysis of passengers of public transport during frontal impact

Kamil Joszko¹, Marek Gzik², Wojciech Wolański³, Bożena Gzik-Zroska⁴, Michał Burkacki⁵, Sławomir Suchoń⁶, Andrzej Muszyński⁷, Karol Zielonka⁸

¹,²,⁴,⁵,⁶ Faculty of Biomechatronics, Silesian University of Technology
Roosevelta 40, 41-800 Zabrze, Poland
e-mail: kamil.joszko@polsl.pl¹, marek.gzik@polsl.pl², bozena.gzikzroska@polsl.pl⁴, michal.burkacki@polsl.pl⁵, slawomir.suchon@polsl.pl⁶

³ Faculty of Biomaterials and Medical Devices Engineering, Silesian University of Technology
Roosevelta 40, 41-800 Zabrze, Poland
e-mail: wojciech.wolanski@polsl.pl

⁷,⁸ Automotive Industry Institute
ul. Jagiellońska 35, 03-301 Warszawa, Poland
e-mail: a.muszyinski@pimot.eu⁷, k.zielonka@pimot.eu⁸

Abstract

Road traffic accidents involving buses do not happen very often, but they are very dangerous, because refers to a large number of passengers. Intercity buses are not equipped with safety belt harness. Therefore, the aim of this study is to analyze results of dynamic loads acting on passengers during frontal impact, who travelling on bus seats with safety belts (two and three-point) and without them. This objective was achieved by experimental studies and modeling, which were focused on process of dynamic load transfer on the human body during traffic accident. The research was conducted in parallel with adult and child. The equivalent of 50 percentile male was Hybrid III dummy, whereas a child age of about 10 years was represented by P10 dummy.

Keywords: bus accidents, multibody, Madymo, safety belts, frontal impact

1. Introduction

Research related to the safety of passengers during traffic accidents are the subject of many publications [1,2]. However, little research was focused on the safety of passengers in buses [3]. Nowadays, the buses are one of the most popular means of public transport in Poland and around the world. In 2013 by means of bus transport, 467 646 000 passengers were transported [5]. However, despite the huge popularity of safety issues improvement during the trip is still insufficient. In 2012 there was 367 accidents in Poland caused by bus drivers in which 17 events were frontal impact. In these accidents 19 people were killed and 632 were injured [5].

The seats in buses are standardized, often there is no possibility of backrest and headrest regulation, they are not also equipped with safety belts. On the basis of knowledge about the genesis of injuries, it can be concluded that public transport should apply passive safety systems in the form of two or three-point safety belt harness. Experimental studies and modeling presented in this paper show what happens to the passengers of public transport in the absence of safety belts and after their application.

2. Materials and methods

Experimental research involved performing frontal impact tests in the physical model similar to the bodywork structure of public transport bus, with dummies located on two seats. The aim of the study was to identify the dynamic loads acting on a human body during a frontal impact. The research was carried out in the Automotive Industry Institute (PIMOT). Design prepared for the research reflects typical intercity bus bodywork (Fig. 1) including three rows of seats of which two were occupied by dummies [4]. The first dummy represented the child about 10 years old (P10) and the second one 50-percentile male Hybrid III. Both of dummies were pinned with a three-point safety belts.

Figure 1: Model setup with dummies

A bus model was placed on trolley devices AB-554. Trolley was accelerated with a rubber rope to the speed of 48 km/h. A desired deceleration was obtained by a special brake. It consists of a trolley attached to the pins which ended with steel knobs (olives). "Olives" and bush made from polyurethane are placed in the tubes restrained to the base of trolley. The braking effect...
occurs through the truck lifting "oil" by polyurethane sleeves. Figure 2 shows the deceleration of the body, measured in the direction of bus movement. For research, implemented deceleration of body was filtered using a CFC60 filter. The model calculations were adopted to the same time step, as recorded during the experimental research (dt=0.0001s).

![Figure 2: Characteristics of bodywork deceleration during the research delay in PIMOT](image)

3. Model implementation

The numerical model was formulated in the Madymo software using the method of multibody (Fig. 3). The dimensions were measured directly on a real object. Positioning of the dummies in seats is described by the characteristic dimensions that define the space around the dummies. Arrangement of the dummies is an important aspect of this project. The arrangement of dummy legs is important for movement of the hips, and therefore on how the lap belt fastens. The following factors were taken into consideration: tilt angle of the upper and lower leg and among others parts, the height of the seat, its length and distance in front of the dummy. Seats were modeled using seven ellipsoids, which represent the characteristic elements in shape of seat, back, head restraint and mounting to the ground. In addition, the model takes into account the floor and one additional child seat in front of a mannequin. The seat and back restraint characteristics is the same in all directions, and is described by the force as a function of displacement. The value of the material properties for the seats selected on the basis of literature data and the same for the whole structure of the seat. Coordinate system in the model coincides with the coordinate system in a real object.

![Figure 3: The model bus with passengers](image)

4. Verification of results

The verification of the model was carried out by comparing the acceleration waveforms recorded for the head and torso. Figure 4 gives an example of the registered acceleration characteristics compared to the dummy's head and forces waveforms for legs and seat belts. In addition, the model was verified by comparing the kinematics from experiment with the model (Fig. 5). With prepared model, several factors influencing safety were considered in tests, for example head acceleration (Fig. 4) and HIC parameters. The simulation results allowed to specify some principles enhancing buses passengers safety in case of traffic accidents.

![Figure 4: Example of the deceleration for the head](image)

![Figure 5: Comparison of kinematics in real object and numerical model](image)

References


Remarks on effective conductivity of nonlinear 2D doubly periodic composites.

David Kapanadze, Gennady Mishuris, Ekaterina Pesetskaya
1,3 A. Razmadze Mathematical Institute, Ivane Javakhishvili Tbilisi State University
Tamarashvili 6, 0177 Tbilisi, Georgia
e-mail: daka@rmi.ge, kate@rmi.ge
2 Department of Mathematics, Aberystwyth University, Ceredigion SY23 3BZ, Wales UK
Rzeszow University of Technology, al. Powstanców Warszawy 12, 35-959, Rzeszów PL
e-mail: ggm@aber.ac.uk

Abstract

The 2D unbounded doubly periodic composite materials are considered, with circular non-overlapping inclusions. The matrix and inclusions are occupied by temperature sensitive materials. A corresponding non-linear boundary value problem is solved analytically in a special case when the conductivities of the matrix and the composite constituents are proportional. As a result, effective conductivity tensor was evaluated and the outcomes compared with results available in literature.

Keywords: temperature dependent doubly-periodic composite, thermal conductivity, effective properties

1. Introduction

Thermal problems related to nonlinear composites can be divided into two classes. The first one appears when the material parameters depend on the gradient of the temperature while the second corresponds to the case when the material parameters are functions of temperature.

Average properties of the composites from the first class were extensively investigated as they are equivalent to the nonlinear dielectrics (see [1] and references thereafter). In contrast, the theory of composite materials with temperature-dependent constituents is not fully developed (at least with respect to applicability of the Hashin-Shtrikman bounds). This work is devoted to a special subset of composites from the second class.

Homogenisation theory for thermal properties of random composites from the second class is studied in [3]. For periodic media, there are several attempts to evaluate the average properties of thermo-sensitive heterogeneous materials. General asymptotic homogenisation techniques for periodic microstructure with temperature independent thermal conductivities were developed in [2]. In [7] and [8], the authors considered the problem for nonlinear composites in terms of Padé approximation. The homogenisation problem was recently revised in [6]. The authors showed that a number of classical methods effective for linear composites (in particular, the Eshelby approach) may be no longer valid for the composites from the second class.

2. Formulation and solution of the BVP

Let us consider non-overlapping disks (inclusions) of different radii located inside the unit square cell \(Q_{(0,0)}\) and periodically repeated in all cells \(Q_{(m_1,m_2)}\) \((m_1,m_2)\) are integer numbers). We search for the temperature and heat flux distribution within the entire composite. The matrix \(D_{\text{matrix}}\) and inclusions \(D_{\text{inc}}\) are occupied by different materials having thermal conductivities \(\lambda = \lambda(T)\) and \(\lambda_k = \lambda_k(T)\), respectively. We assume that the conductivities \(\lambda, \lambda_k \) \((k = 1, \ldots, N)\) are continuous bounded strictly positive real functions.

The temperature \(T = T(x, y)\) satisfies the nonlinear partial differential equations:

\[
\nabla(\lambda(T) \nabla T) = 0, \quad (x, y) \in D_{\text{matrix}},
\]

\[
\nabla(\lambda_k(T) \nabla T) = 0, \quad (x, y) \in D_{\text{inc}}.
\]

We assume that the perfect (ideal) contact conditions are valid along the boundaries between the matrix and inclusions:

\[
T(s) = T_k(s), \quad \lambda(T(s)) \frac{dT(s)}{dn} = \lambda_k(T_k(s)) \frac{dT_k(s)}{dn}.
\]

Average flux vector of intensity \(A\) is prescribed and directed at an angle \(\theta\) to axis \(Ox\) which does not coincide, in general, with the orientation of the periodic cell. This gives the following conditions:

\[
\int_{Q_{(m_1,m_2)}} \lambda(T) T_y \, ds = -A \sin \theta, \quad \int_{Q_{(m_1,m_2)}} \lambda(T) T_x \, ds = -A \cos \theta.
\]

Since there are no heat sources and sinks in the composite, conditions (4) can be replaced with those defined on the opposite sides of the cell.
In order to solve the problem we use the Kirchhoff transformations
\[ f(T) = \int_0^T \lambda(\xi) \, d\xi, \quad f_k(T) = \int_0^T \lambda_k(\xi) \, d\xi. \]
This leads us to an equivalent problem for the functions \( u(x, y) = f(T(x, y)), \ u_k(x, y) = f_k(T_k(x, y)) \) satisfying Laplace equations
\[ \Delta u = 0, \ (x, y) \in D_{\text{matrix}}, \ \Delta u_k = 0, \ (x, y) \in D_{\text{inc}}, \]
with nonlinear transmission conditions
\[ u = f(f_k^{-1}(u_k)), \quad \frac{\partial u}{\partial n} = \frac{\partial u_k}{\partial n}. \]

As mentioned above, we assume that the conductivities are not arbitrary satisfying the following condition
\[ \lambda(T) = C_k \lambda_k(T), \]
where \( C_k \) are known constants. As a result, the problem (5), (6) was reduced to a linear one, since the derivatives are directly computed to give (for more details see [5]):
\[ (f(f_k^{-1}(\xi)))' = \frac{\lambda(T)}{\lambda_k(T)} \xi = f_k(T_k). \]

3. Effective properties of the composite

The effective conductivity tensor, \( \Lambda_e \), is investigated depending on average temperature \((T)\) and defined in the following way:
\[ \langle \lambda(T) \nabla T \rangle = \Lambda_e \langle \nabla T \rangle. \]

Two numerical procedures were used to evaluate the effective properties of the composite.

- First, the auxiliary linear boundary value problem is solved in a doubly periodic domain preserving its uniqueness by any appropriately chosen condition. Then, we compute the component of the effective conductivity tensor for each particular unit cell. This defines the material properties in a discrete way (where the numbers of the points is infinite though).

- Another method to evaluate the effective properties was to focus on one (arbitrary chosen from the original domain) elementary cell only and to build a set of solutions to the auxiliary linear problem in the form \( u = u_e + C \), where \( C \) is an arbitrary constant. Then, for every constant \( C \), the components of the effective conductivity tensor, \( \Lambda_e \), and the average temperature, \( \langle T \rangle \), are functions of the parameter \( C \). Changing the value of \( C \) continuously from \(-\infty \) to \( \infty \), the effective conductivity tensor is stated implicitly as a continuous function of the average temperature. It is clear that this procedure does not depend on the choice of the cell.

Both of these approaches were used in computations (see [5] for details). This allows us to reconstruct the effective properties of the composite with the nonlinear properties and to compare our results with those from [6] for random composites. We also discuss the basic Reuss-Voigt type estimations and the Hashin-Shtrikman bounds for the considered composites following the results from [2] and show limitations for their validity.

Acknowledgements. GM is grateful for support from the FP7 IRSES Marie Curie grant TAMER No 610547. DK and EP are supported by Shota Rustaveli National Science Foundation with the grant number FR/6/S-101/12 31/39.

References


Notes on the Mechanics of the Octopus’s Arm

Dorian Kim\textsuperscript{1}, Reuven Segev\textsuperscript{2}\textsuperscript{*}

\textsuperscript{1,2} Department of Mechanical Engineering, Ben-Gurion University of the Negev
P.O.Box 653, Beer-Sheva 84105, Israel
e-mail: rsegev@bgu.ac.il

Abstract

Various issues raised by the mechanics of an octopus’s arm are presented from the point of view of continuum mechanics. First, we study the conditions for a mixture of a number of fiber bundles to span the space of stress tensors. The geometry of a fiber bundle, as reflected in the direction and density of the fibers, is described by a vector field, or more precisely, by a differential two-form. For the sake of stress analysis we propose a three-dimensional continuum model of the arm that is based on constitutive data available for muscle fibers. An analysis of small deformations superimposed on an activated configuration of the muscle bundles reveals the stiffening mechanism which enables the arm to support various external loadings, even if the geometry and the number of the fiber groups does not allow the tensions in them to span the space of stress tensors.

Keywords: octopus, continuum mechanics, stresses, muscular hydrostat, activated fibers, incompressibility

1. Introduction

The structure of the arm of an octopus enables dynamic and static control of the arm, compensating for the inability of muscles to extend actively and for the absence of a rigid skeleton. The prospect of mimicking the structure of the arm as a mechanical soft manipulator using active materials as actuators, motivates the interest in studying the morphology, kinematics and mechanics of the arm.

The arm of an octopus contains five groups of muscle fibers: longitudinal fibers, two groups of mutually orthogonal transverse fibers, and two groups of right handed and left handed helicoidal fibers (see Figure 1). When the muscles are inactive, the material that makes up the arm does not support shear statically. In addition, the material shows very little compressibility which is therefore usually neglected. Such a structure is referred to as a muscular hydrostat. The elephant trunk and the human tongue are other examples of organs that have the structures of muscular hydrostats. (See a detailed study in Kier and Stella, [1].)

In a study of the mechanics of an octopus’s arm, it is desirable to explain how the arm operates without having a rigid skeleton and in spite of the fact that muscle fibers can apply tension and cannot support compression. As an elementary example, due to incompressibility, the arm can extend and apply longitudinal compression by contracting the transverse muscle groups (see illustration in Figure 2). Similarly, Figure 3 gives a schematic illustration of the way the arm bends.

Consider also the pitch angle $\theta_m$ of the oblique muscles for which the length of the helicoidal fibers will be minimal. In such a case, both extension of the arm and its contraction will cause the helicoidal fibers to extend. Thus, tension in the helicoidal fibers will resist both an extension and a contraction of the arm without activating either the longitudinal muscles or the transverse fibers. It turns out that the computed angle is the “magic angle” satisfying $\cos \theta_m = \sqrt{3}$ which emerges time and again in mathematics and physics.

The following present our analyses of various issues related to the study of the continuum mechanics of the octopus’s arm.

2. Fiber Bundles and Bases for the Space of Tensors

It is assumed throughout that five distributed intertwined muscle fibers groups are embedded in the arm and that the total mixture is incompressible.
The structure of the arm raises some theoretical questions. Since the space of the values of the stress tensor at a point, the space of $3 \times 3$ symmetric tensors, is 6-dimensional, the question arises whether it can be spanned by a collection of 6 stress tensors corresponding to tensions in 6 fiber bundles. In other words, we search for the conditions such that for 6 vectors, $w_m$, $n = 1, \ldots, 6$, the tensors $w_m \otimes w_m$ be linearly independent. Since the fibers may be only in tension, the collection of stresses resulting from tension in the fibers (rather than compression) is investigated. Next, taking incompressibility into consideration, conditions are sought for the geometry of a collection of five vectors, $w_m$, $m = 1, \ldots, 5$, so that the tensors $w_m \otimes w_m$ together with the identity tensor $I$ span the space of symmetric tensors. For example, we show that a necessary condition for the tensors $w_m \otimes w_m$ to be linearly independent is that there are no triplets of mutually perpendicular vectors. Another sufficient condition is that the five vectors are not situated in two perpendicular planes. In fact, if the three vectors are situated in one plane and the other two vectors are situated in another, non-orthogonal plane, the resulting tensors are linearly independent, i.e., this is a sufficient condition.

In fact, from a general geometric point of view, the fiber density is better described as a differential 2-form $\omega$, an anti-symmetric 2-tensor. The differential form $\omega$ is independent of the configuration of the fiber bundle, particularly, independent of $F$ and it is related to the vector field $w$ by

$$\omega(v, u) = w \cdot (v \times u)$$

for any two vectors $v, u$. Equation (2) may be rewritten in terms of $\omega$ using the exterior derivative operator $d$ as

$$d\omega = 0.$$  

(5)

It is observed that for the case of an incompressible bundle of fibers, as one assumes for the octopus’s arm, $|F| = 1$, and so the fiber density field transforms as a genuine vector field.

Let us denote the tension in a single muscle fiber by $T$. Then, the Cauchy stress in the muscle bundle is given by

$$\sigma = T \hat{w} \otimes w,$$

where $\hat{w}$ denotes the unit vector in the direction of $w$. Similarly, it is easy to show that the first and second Piola-Kirchhoff stress tensors are given respectively by

$$P = T \hat{w} \otimes W,$$  

$$S = \frac{T}{|w|} W \otimes W = \frac{T}{\lambda} \hat{W} \otimes W,$$

where $\lambda = |w|/|W| = |F(W)|/|W|$ is the extension and the incompressibility constraint was used.

It is assumed that there is no interaction between the various muscle bundles of the arm and it follows that the total Cauchy stress is given by

$$\sigma = \sum_{m=1}^{5} T_m \hat{w}_m \otimes w_m - pI.$$  

(8)

The equilibrium equations assume a particularly simple form if one assumes that the tension in the fibers and the extension of the fibers are uniform along the fibers and that the density of the fibers is uniform in the reference state. Under these assumptions one has

$$\sum_{m=1}^{5} T_m \hat{w}_m \cdot \text{grad} \, \hat{w}_m - \text{grad} \, p + b = 0,$$

where $b$ denotes the body forces.

4. The Formulation of the Stress Analysis Problem

For the stress analysis of the arm, a constitutive model of the muscle fibers should be incorporated. In order to make use of available data on the mechanical properties of muscle fibers, we conceive the following scenario. From a reference configuration which is not activated, the arm is deformed to an intermediate configuration in an unloaded motion for which the activation of the muscles is negligible. At this intermediate configuration, the muscles are activated isometrically. Then, under fixed activation of the muscles, the arm is loaded externally undergoing a passive small deformation which is superimposed on the foregoing large deformation. Linearization of the constitutive relations in a neighborhood of the prestressed activated muscles shows how geometric stiffness enables the arm to support external loading even with a smaller number of muscle groups.

References

Sensitivity analysis of temperature field in the heated tissue with respect to the dual-phase-lag model parameters

Ewa Majchrzak1,*, Łukasz Turchan1,*, Grażyna Kaluza2

1 Institute of Computational Mechanics and Engineering
Silesian University of Technology, Konarskiego 18A, 44-100 Gliwice, Poland

e-mail: ewa.majchrzak@polsl.pl 1, lukasz.turchan@polsl.pl 2, grazyna.kaluza@polsl.pl 3

Abstract

Heat transfer processes in axially symmetrical domains of heated tissue are analyzed. The problem is described by a dual-phase-lag equation (DPLE) supplemented by appropriate boundary and initial conditions. The aim of the research is to estimate the variations in the transient temperature field due to the perturbations of the DPLE thermophysical parameters. The methods of sensitivity analysis are applied for the task. The direct problem and additional ones connected with the sensitivity analysis are solved using the explicit scheme of finite difference method. In the final part of the paper computational examples are presented.

Keywords: bioheat transfer, dual-phase-lag equation, sensitivity analysis, finite difference method

1. Introduction

Heat transfer processes taking place in the heated tissues can be described by different models [5]. In the paper the dual-phase-lag equation is considered [6]. This model allows to take into account non-homogeneous structure of a biological tissue. Thermophysical parameters occurring in the dual-phase-lag equation differ significantly from person to person. Thus, it is important to assess the impact of thermophysical parameters perturbations on the course of thermal processes in the considered domain. The method of sensitivity analysis [1, 2] was chosen here. The governing equations are differentiated with respect to the parameters, thus additional problems are formulated. The basic problem and additional ones are solved using the finite difference method. In the final part the results of computations are shown.

2. Governing equations

Dual-phase-lag equation in a cylindrical coordinate system reads [6]

\[
\begin{align*}
\left[ \frac{\partial T(r,z,t)}{\partial t} + \tau_q \frac{\partial^2 T(r,z,t)}{\partial t^2} \right] & = \lambda \nabla^2 T(r,z,t) + \\
\lambda \tau_q \frac{\partial \nabla^2 T(r,z,t)}{\partial t} + \frac{\partial Q(r,z,t)}{\partial t} & + \tau_e \frac{\partial Q(r,z,t)}{\partial t} = 0
\end{align*}
\]

(1)

where \( c \) is the volumetric specific heat of tissue, \( \lambda \) is the thermal conductivity of tissue, \( T \) is the temperature, \((r,z)\) are the spatial coordinates, \( t \) is the time and \( Q(r,z,t) \) is the source term due to metabolism and blood perfusion. In equation (1) \( \tau_q \) is the relaxation time and \( \tau_T \) is the thermalization time.

The source term \( Q(r,z,t) \) can be written in the form

\[
Q(r,z,t) = w_B c_B \left[ T_B - T(r,z,t) \right] + Q_m
\]

where \( w_B \) is the blood perfusion rate, \( c_B \) is the specific heat of blood, \( T_B \) is the blood temperature and \( Q_m \) is the metabolic heat source.

The equation (1) is supplemented by the boundary conditions (Figure 1)

\[
\begin{align*}
(r,z) & \in \Gamma_1: \quad T = T_b \\
(r,z) & \in \Gamma_2: \quad q(r,z,t) = q_0(r,z,t) \\
(r,z) & \in \Gamma_3: \quad q(r,z,t) = 0
\end{align*}
\]

(3)

and the initial ones

\[
t = 0: \quad T = T_p, \quad \frac{\partial T}{\partial t} \bigg|_{t=0} = 0
\]

(4)

where \( T_b \) is the known boundary temperature, \( q_0 \) is the known boundary heat flux and \( T_p \) is a constant initial tissue temperature.

For \( r \leq r_0, \quad z = 0 \) and \( t \leq t_e \), where \( t_e \) is the exposure time, the Neumann boundary condition is accepted [4]

\[
q_0(r,0,t) = q_0 \left[ 1 - \frac{t}{t_e} \right] \exp \left( - \frac{r^2}{r_0^2} \right)
\]

(5)

where \( q_0 \) is the constant value, for \( t > t_e: \quad q_0(r,0,t) = 0 \).

It should be pointed out that using the dual phase lag model the following form of a second type boundary condition should be considered

\[
(r,z) \in \Gamma_2: \quad q_0 = -\lambda \left( \frac{\partial T}{\partial n} + \tau_T \frac{\partial T}{\partial n} \right)
\]

(6)

where \( n \) is the normal outward vector and \( \partial T/\partial n \) is the normal derivative.
3. Sensitivity analysis

Let $p_1 = \lambda$, $p_2 = c$, $p_3 = T_B$, $p_4 = w_B$, $p_5 = Q_m$. The equation (1) is differentiated with respect to the parameter $p_s$, $s = 1, 2, 3, 4, 5$ and then

$$
\frac{c + \tau w_B c}{\partial t} + \tau c \frac{\partial^2 U}{\partial t^2} = \lambda c^2 U_s + \tau c \frac{\partial U}{\partial t} - w_B c U_s +
$$

$$
\left[ \frac{\partial \lambda}{\partial p_s} c + \frac{\partial c}{\partial p_s} \right] \frac{\partial U}{\partial t} + \frac{\partial w_B}{\partial p_s} c \frac{\partial U}{\partial t} - \frac{\partial c}{\partial p_s} \frac{\partial U}{\partial t} = \frac{\partial T}{\partial p_s} + \lambda c \frac{\partial^2 U}{\partial t^2} +
$$

$$
\left[ \frac{\partial \lambda}{\partial p_s} c - \frac{\partial \lambda}{\partial p} c \right] \frac{\partial T}{\partial p_s} + \frac{\partial c}{\partial p_s} \frac{\partial T}{\partial p_s} = \frac{\partial T}{\partial p_s} + \lambda c \frac{\partial^2 T}{\partial t^2} +
$$

$$
\frac{\partial w_B}{\partial p_s} c_s (T_B - T) + \frac{\partial Q_m}{\partial p_s} = \frac{1}{\lambda c} \left[ w_B c_s (T_B - T) + Q_m \right]
$$

where $U_s = \frac{\partial T}{\partial p_s}$ is the sensitivity function. The boundary conditions (3) and the initial ones (4) are also differentiated with respect to $p_s$. Thus

$$(r, z) \in \Gamma_s : U_s = 0
$$

$$(r, z) \in \Gamma_s : - \frac{\partial U_s}{\partial n} + \lambda c \frac{\partial \lambda c}{\partial n} + \tau c \frac{\partial U}{\partial n} + \frac{\partial \lambda c}{\partial n} \frac{\partial \lambda c}{\partial n} = 0
$$

$$(r, z) \in \Gamma_s : \lambda c \frac{\partial U_s}{\partial n} + \frac{\partial T}{\partial p_s} = - \lambda c \frac{\partial U_s}{\partial n} + \frac{\partial T}{\partial p_s} = 0
$$

and

$$
t = 0 : U_s = 0, \frac{\partial U_s}{\partial t} \bigg|_{t=0} = 0
$$

In this way the additional problems connected with the sensitivity functions $U_s$ are formulated.

4. Results of computations

The cylindrical tissue domain ($R=0.02$ m, $Z=0.02$ m) of the initial temperature $T_p = 37^\circ$C is considered. The following values of parameters are assumed: thermal conductivity of tissue $\lambda = 0.5$ W/(mK), volumetric specific heat $c = 4$ MJ/(m$^3$K), blood perfusion rate $w_B = 0.53$ kg/(m$^3$s), specific heat of blood $c_B = 3770$ J/(kgK), blood temperature $T_B = 37^\circ$C, metabolic heat source $Q_m = 250$ W/m$^3$, relaxation time $\tau_q = 15$ s, thermalization time $\tau_T = 10$ s.

In boundary condition (5): $q_0 = 2000$ W/m$^2$, $r_D = R/4$, $t_e = 100$ s. The problem is solved using explicit scheme of finite difference method (time step: $\Delta t = 0.0005$ s, grid step: $h = 0.0002$ m) [3].

In Fig. 2 the temperature history at the points A(0,0), B(0, 10h) and C(0, 20h) is shown, while Figs 3 and 4 illustrate the courses of sensitivity functions $U_3 = \frac{\partial T}{\partial \tau_q}$ and $U_4 = \frac{\partial T}{\partial \tau_T}$ at the same points.

The sensitivity functions allow to estimate the changes of temperature distribution due to the perturbations of parameters. These problems will be discussed in the full version of the paper.

References


1. Introduction

We discuss 2D two-phase fiber composites represented by a section displayed in Fig. 1. Unidirectional fibers of a finite conductivity are embedded in the host material of another conductivity. Besides the classic problems of the effective conductivity we consider the RVE theory and its applications to the stir casting process in the 2D statement. The presented results can be considered as a constructive application of the stochastic homogenization [7].

Microstructure of composites and its dynamic changes depend on geometrical parameters. A proper choice of these parameters and a possibility of their effective computations can give a description of the corresponding local physical fields and of the macroscopic properties of composites. In this paper, we propose the following choice of the geometric parameters describing random structures

\[ G = \{e_p, p \in \mathcal{M}_L\} \] (1)

where the set \( \mathcal{M}_L \) is precisely defined in [5] and the elements \( e_p \) are defined in following section.

2. General theory of basic sums

We follow [3] to introduce basic sums. Let \( a_k \ (k = 1, 2, \ldots, N) \) be a set of points. Let \( q \) be a positive integer; \( k_i \) runs over 1 to \( N \); \( m_j = 2, 3, \ldots \). Let \( C \) be the operator of complex conjugation. Introduce the following sum of multi-index \((m_1, \ldots, m_q)\)

\[ e_{m_1 \ldots m_q} := N^{-\left[1+\frac{1}{2}(m_1+\cdots+m_q)\right]} \sum_{k_0 \cdots k_q} E_{m_1}(a_{k_0} - a_{k_1}) \times E_{m_2}(a_{k_1} - a_{k_2}) \cdots C^{q+1} E_{m_q}(a_{k_{q-1}} - a_{k_q}). \] (2)

Sums \( e_m \) becomes the Eisenstein-Rayleigh lattice sums \( S_m \) [3] in the case \( N = 1 \) since it is assumed for convenience that

\[ E_m(0) := S_m. \] (3)

The Eisenstein functions \([6]\) are related to the Weierstrass function \( \wp(z) \) \([1]\) by the identities

\[ E_2(z) = \wp(z) + S_2, \] (4)

\[ E_m(z) = \frac{(-1)^m}{(m-1)!} \frac{d^{m-2}\wp(z)}{dz^{m-2}}, \quad m = 3, 4, \ldots \] (5)

Every function (5) is doubly periodic and has a pole of order \( m \) at \( z = 0 \). The sums 2 constitute the basic elements to calculate the effective conductivity \([4]\) dependent only on the locations of inclusion.

3. Classification of the basic sums

3.1. Description of non-overlapping distributions

Let \( N \) disks of radius \( r \) be uniformly distributed in doubly periodic cell \( Q_{(0,0)} \) (see Fig. 1) without mutual overlapping and let \( \nu = N \pi r^2 \) denote their concentration in the unit cell.

![Figure 1: Doubly periodic composite with inclusions](image)

The centers of the disks \( a_k \ (k = 1, 2, \ldots, N) \) can be considered a probabilistics event when a set of independent and identically distributed (i.i.d.) random points are located in all possible unit cells \( Q_{(0,0)} \). Let \( \mathcal{U}_v(0) \) denote a location of disks at the ini-
monotonically \cite{5}. Figure 3 demonstrates that $e_{m}$ can be certainly distinguished for $t < 300$.

Figure 3: The sum $e_{4,4}$ against the random walk time $t$. Data for the initial locations: square lattice (black dots); hexagonal lattice (grey dots). The concentration holds $\nu = 0.7$

Typical dependencies of the basic sums on the concentration are presented in Fig. 4 for $0.2 \leq \nu \leq 0.75$. It is observed that the sums either slightly oscillate around certain value or converge monotonically \cite{5}.

Besides $\nu$ and $t$, the sums $e_{m}$ must depend numerically on $N$. Such a large number $N$ has to be taken that $e_{m}$ weakly depends on $N$. Another type of classification of the basic sums is related to the effective properties of composites (see \cite{5}).

3.3. Application to composites

The main interest in the study of basic sums is related to computation of the effective conductivity of random 2D composites and to a simple computational algorithm to determine the RVE was described in \cite{3}. Authors of \cite{5} demonstrated that $e_{m}(t)$ can be applied to the stir casting process and to the mysterious effect of the memory in stirring. Preliminary useful applications to the stir casting process were described in \cite{2}.

These basic sums perform the same role in description of microstructure as the $n$-point correlation functions. The main advantage of $e_{m}$ is their fast computation including high order terms.

References

\begin{itemize}
\end{itemize}
Numerical modeling of biological tissue freezing process using the dual-phase-lag equation

Bohdan Mochnacki1, Ewa Majchrzak2*

1 Chair of Mathematics and Computer Science
Higher School of Labour Safety Management, Bankowa 8, 40-007 Katowice, Poland
e-mail: bmochnacki@wszop.edu.pl

2 Institute of Computational Mechanics and Engineering
Silesian University of Technology, Konarskiego 18A, 44-100 Gliwice, Poland
e-mail: ewa.majchrzak@polsl.pl

Abstract

Heat transfer processes proceeding during the biological tissue freezing are discussed. The problem is described by dual-phase lag equation (DPLE) supplemented by appropriate boundary and initial conditions. The freezing models so far presented based on the well known Pennes equation, but taking into account the physical properties of tissue the DPLE seems to be the better approximation of thermal processes taking place in the subdomain of being frozen tissue. The characteristic feature of DPLE is the introduction of two lag times called the relaxation and thermalization times \(\tau_r\) and \(\tau_t\). The cryosurgery treatment using the spherical internal cryoprobe is considered. At the stage of numerical computations the explicit scheme of FDM for nonlinear problems is applied. The example of simulation is also presented.

Keywords: bioheat transfer, tissue freezing, dual-phase-lag equation, numerical methods, finite difference method

1. Introduction

The typical description of the bioheat transfer processes (the tissue models [1,2,4,5]) is based on the well known Pennes equation supplemented by the appropriate boundary and initial conditions, but recently starts to dominate the view that taking into account the tissue properties, the model using DPLE is the better one. The specific, non-homogeneous structure of the tissue causes that the introduction of lag times is entirely justified [6].

The biological tissue freezing process proceeds in the interval of temperature (from \(-1^\circ C\) to \(-8^\circ C\)). The evolution of the freezing heat and the cooling process in the intermediate region can be taken into account by the introduction of a parameter called an effective thermal capacity [1,3]. Obtained in this way governing equation corresponds to the one domain method [1,3], because the only one equation describes the thermal processes in the whole, conventionally homogeneous tissue domain. The problem has been solved using a variant of the finite difference method presented in [2]. In the final part the results of computations are shown.

2. Governing equations

The following energy equation is considered [6]

\[
\frac{\partial T}{\partial t} + \tau_r \frac{\partial T}{\partial t} + \tau_t \frac{\partial T}{\partial t} = \nabla (\lambda(T)\nabla T) + \nabla \left( \frac{\nabla (\lambda(T)\nabla T)}{\partial t} \right) + \frac{Q}{\partial t} + \tau_r \frac{\partial Q}{\partial t} + \tau_t \frac{\partial Q}{\partial t}
\]

So, the source term \(Q(x, t)\) is the sum of the following components

\[
Q(x, t) = w_s(T) c_s(T) (T - T_s) + Q_m(T) + L \frac{\partial S(x, t)}{\partial t}
\]

where \(w_s(T)\) is the blood perfusion rate, \(c_s\) is the specific heat of blood, \(T_s\) is the blood temperature, \(Q_m\) is the metabolic heat source, \(L\) is the volumetric freezing heat and \(S(x, t)\) is the frozen state fraction at the neighborhood of the point considered.

It should be pointed out that the blood perfusion rate and the metabolic heat source are equal to zero for the frozen region, while for the intermediate one the linear changes starting from the point \(-1^\circ C\) corresponding to the natural state of tissue are assumed.

The mathematical manipulations typical for the case of one domain approach lead to the energy equation in the form

\[
\frac{C(T) + \tau_r w_s(T)c_s(T) + \tau_t C(T)}{\partial t} = \nabla \left( \frac{\nabla (\lambda(T)\nabla T)}{\partial t} \right) + \frac{w_s c_s(T_s - T)}{\partial t} + Q_m(T)
\]

where \(C(T)\) is the effective thermal capacity, at the same time for natural state of tissue and for the frozen region this parameter corresponds directly to the volumetric specific heats of these sub-domains, while for the intermediate region it has a form of bell-type function shown in Fig. 1 [1].

The course of thermal conductivity was approximated by the continuous and differentiable function created by the constant values and the polynomial of the third degree as shown in Fig. 2.

The action of spherical cryoprobe tip causes the generation of time-dependent ice ball and the problem has been treated as 1D one for the object oriented in the spherical co-ordinate system. The example presented in the final part of the paper concerns the constant value of cryoprobe tip temperature (the Dirichlet condition) but the time-dependent ice ball surface temperature can be also introduced. On the surface limiting the tissue domain the no-flux condition is assumed.

The paper was prepared as a part of project PB3 sponsored by Higher School of Labour Safety Management
Additionally, for $t = 0$:

$$ t = 0; \quad T = T_p, \quad \frac{\partial T}{\partial t} \bigg|_{t=0} = 0 $$

(4)

where $T_p$ is the initial tissue temperature.

The spherical tissue domain of radius $R=0.015\text{m}$ is subjected to the cryoprobe of radius $R_1=0.005\text{ m}$. The initial temperature of tissue equals $T_p=37{^\circ}\text{C}$. The following values of parameters are assumed: blood perfusion rate and metabolic heat source for natural state ($T>−1{^\circ}\text{C}$) $w_B=0.53\text{ kg/(m^3s)}$ and $Q_m=250\text{ W/m}^3$, respectively, specific heat of blood $c_B=3770\text{ J/(kgK)}$, blood temperature $T_B=37{^\circ}\text{C}$, relaxation time $\tau_q=5\text{ s}$, thermalization time $\tau_T=1\text{ s}$. Additionally, the cryoprobe tip temperature is equal to $−90{^\circ}\text{C}$.

The problem is solved using explicit scheme of finite difference method for nonlinear hyperbolic equations (time step: $\Delta t=0.001\text{ s}$, grid step: $h=0.00005\text{ m}$).

In Figure 3 the temperature distribution in domain of tissue is shown, while Fig. 4 illustrates the cooling curves at the points A, B and C located at the distance 0.001 m, 0.002 m and 0.003 m from the cryoprobe surface.

The algorithm elaborated can be used for the more complex problems solution (e.g. geometry of tissue domain, time-dependent cryoprobe tip temperature, etc.).

3. Results of computations

The algorithm elaborated can be used for the more complex problems solution (e.g. geometry of tissue domain, time-dependent cryoprobe tip temperature, etc.).

References


Modified Hu-Washizu principle as a general basis for FEM plasticity equations

Kazimierz Myślecki¹, Jakub Lewandowski²

¹,² Faculty of Civil Engineering, Wroclaw University of Technology
11 Grunwaldzki Sq., 50-377 Wroclaw, Poland
e-mail: kazimierz.myslecki@pwr.edu.pl¹, jakub.lewandowski@pwr.edu.pl²

Abstract

The modified Hu-Washizu variational principle is a general approach to solve various plasticity problems. It makes it possible to derive FEM equations for any yield surface formulation (including a hardening model) and any element (regarding dimensions and shape functions). A step-by-step procedure to obtain specific FEM equations is shown. Verification on a simple problem is done. Two examples (plane stress triangular FEs, cantilever beam, different yield formulae) illustrate a work of the FEM algorithm which is based on the modified Hu-Washizu variational principle.

Keywords: FEM, plasticity, hardening, variational principle, functional, Hu-Washizu

1. Introduction

The FEM equations may be derived from different variational principles. Each one shows a specific application range and can be modified to fulfill additional constraints. The original Hu-Washizu variational principle [1] is used for elastostatics. It is also often modified to be applied to plasticity problems as well. However, usually its modifications are designed to describe a certain case, e.g. plane stress plasticity with kinematic hardening for triangular elements. No universal approach exists to be used for any plasticity model and for any FE definition. The authors propose a general modified version of the principle which fulfills these aims:

\[ \Pi(\sigma_0 - \sigma_0^p) + \Delta F_e = 0 \]

where respective symbols designate:

- \( C_{ijkl} \) - stiffness tensor
- \( \sigma_{ij} \) - stress and strain tensors
- \( \epsilon_{ij} \) - plastic strain tensor
- \( u_i \) - displacement vector
- \( \Delta F_e \) - increment of body forces vector

The omitted part (‘...’) are boundary conditions which are provided in FEM by the stiffness matrix modification.

The functional stationarity satisfies the associated flow rule incremental equations:

\[ \begin{align*}
(\sigma_0 - \sigma_0^p)_{ij} + \Delta F_e &= 0 \\
\frac{1}{2} (u_{rj} + u_{jr}) &= \epsilon_{ij} + \epsilon_{ij}^p + \lambda \frac{\partial \Phi(\sigma_0)}{\partial \sigma_0} \\
(\sigma_0 - \sigma_0^p) &= C_{ijkl} (\epsilon_{ij} - \epsilon_{ij}^p) \\
\Phi(\sigma_0) &= 0
\end{align*} \]

The incremental form is essential for nonlinear problems, such as plasticity. The index "\( h \)" corresponds to the values calculated in the previous incremental step whereas no index means the current value. \( \Phi \) may be any yield function. Hence, it is possible to derive FEM equations for any yield surface. It may be an object of future studies to find a variational principle for different flow rules.

2. FEM equations and algorithm

For a specific FE, assuming a constant deformation matrix (B), the functional has a discrete form:

\[ \Pi(\sigma, \epsilon, q, \lambda) = \frac{1}{2} (\epsilon - \epsilon^p)^T C (\epsilon - \epsilon^p) - V^T \Delta \Phi \]

\[ - (\sigma - \sigma^p)^T \left( \epsilon + \epsilon^p - B^T q \right) V - \lambda \Delta \Phi V, \]

where \( V \) is the size of the element (length, area, volume). Symbols are analogous to Eqn (2). Capital letters (\( C, B \)) are reserved for matrices and small letters (\( \sigma, \epsilon, q, \Delta \)) – for vectors.

The FEM equations are obtained by the stationarity of the functional. They can be expressed as:

\[ \begin{bmatrix}
0 & B^T V & 0 & q \\
0 & C^T V & -1V & 0 \\
B^T & -IV & K^{NN}_s & K^{NI}_s \\
0 & 0 & K^{SN}_s & K^{II}_s
\end{bmatrix} \begin{bmatrix}
\sigma \\
\epsilon \\
\epsilon^p \\
\lambda
\end{bmatrix} = \begin{bmatrix}
\Delta \Phi + B^T \sigma^p V \\
0 \\
C \epsilon^p - 1V \\
0
\end{bmatrix} \]

where \( K^{NN}_s, K^{NI}_s, K^{SN}_s, K^{II}_s \) and \( b^s \) depend on the yield surface equation \( \Phi \) (with or without hardening). Equation (4) is complete. The variational principle (Eqn (1)) provides that it contains all plasticity formulae and no additional equations are needed. It is nonlinear and solved with the Newton-Raphson method. Considering the block form of the Eqn (4) and its zero submatrices, it is possible to reduce the FEM algorithm computation time by block elimination. The algorithm was implemented in Mathematica.
The return mapping method tentatively assumes elastic behavior of the element throughout a given time step. If the resulting stress violates the yield condition, it is projected back to the plastic yield surface by enforcing Eqn (4) where \( \sigma^e \) and \( \varepsilon^e \) are results from the “elastic prediction”, as illustrated below:

![Figure 1. The return mapping algorithm](image)

3. **Examples**

To show the efficiency of the algorithm (based on Eqn (4)), two examples were solved. In each case, the problem is a cantilever beam, built of plane stress triangular FEs, with the same load program. \( C \) and \( B \) are commonly used stiffness and deformation matrices for such FEs [2]. The examples differ only by hardening model for the Huber-Mises-Hencky yield criterion.

3.1. **Ideal plasticity**

The first example deals with ideal plasticity without hardening. In this case the yield function can be easily expressed as a function of stress vector:

\[
\Phi(\sigma) = \frac{1}{2} \sigma^T \psi \sigma - 2\sigma^T_0 \psi \psi = \begin{bmatrix} 4 & -2 & 0 \\ -2 & 4 & 0 \\ 0 & 0 & 12 \end{bmatrix},
\]

where \( \sigma^e_0 \) is a yield stress referring to the HMH reduced stress. \( \Phi(\sigma) \) implies matrices definitions for Eqn (4):

\[
\begin{align*}
K_{SS}^{\psi} &= -\psi \sigma V, \\
K_{S}^{\psi V} &= -\frac{1}{2} \sigma^T \psi V, \\
k_{S}^{\psi V} &= K_{S}^{\psi V} = 0, \\
b^T &= -2\sigma^T V
\end{align*}
\]

3.2. **Kinematic hardening**

The second example is the HMH yield criterion with simple kinematic hardening. Assuming a proportional stiffness \( \alpha C \) \( (0 < \alpha < 1) \) after yield stress exceeding, the yield surface formula is

\[
\Phi(\sigma, \varepsilon) = \frac{1}{2} \sigma^T \psi \sigma - \frac{\alpha}{1-\alpha} \sigma^T \psi C \varepsilon
\]

\[
+ \frac{1}{2} \frac{\alpha}{1-\alpha} \varepsilon \psi \varepsilon C \varepsilon - 2\sigma^T_0 \psi \psi.
\]

It depends on the stress vector as well as on the current plastic strain vector \( \varepsilon^e \). In this case submatrices in Eqn (4) are more expanded than for the ideal plasticity (Eqn (6)).

4. **Verification**

The authors verified their algorithm on the simple example:

![Figure 2. Plate under plane stress](image)

To show the influence of kinematic hardening during compression and tension, proper loading program is applied (Fig. 3). Two cycles of pressure change are included.

![Figure 3. Loading program](image)

The Young’s modulus, dimensions and yield stress are adjusted so that pressure value strongly exceeds the elastic range \( \max \sigma = 1.5 \sigma^e \). The HMH yield criterion with kinematic hardening (Eqn (7)) is used. Assuming value of the hardening coefficient \( \alpha = 0.1 \), the algorithm gives results as shown below:

![Figure 4. Loading-displacement relation](image)

The solution (Fig. 4) is almost the same as the analytical one for the equivalent 1D beam. Two loading cycles cover each other in the pressure-displacement plane. The \( p(u) \) line inclination in the plastic flow range is 0.1 of the initial value.

References


Simulation of indentation test for lymphedematous tissue within poroelastic model

Joanna Nowak¹, Mariusz Kaczmarek²

¹,² Institute of Mechanics and Applied Computer Science, Kazimierz Wielki University
Kopernika 1, 85-074 Bydgoszcz, Poland
e-mail: joanna_n@ukw.edu.pl

Abstract

The indentation test is the most commonly used method of assessing mechanical properties of soft biological tissues. Especially in the cases of lymphedematous tissue it is an excellent alternative for the standard “palpation test”. The indentation test may allow for evaluation of a presence or stage of edema as well as for assessment of evolution of properties of tissues due to therapy (e.g. as reaction to manual or pneumatic massage). This work presents results of numerical simulations for a simplified cylindrical geometry of tissue considered as two-phase fluid saturated porous medium compressed by a flat-ended indenter. The assumptions of hyperelastic material of skeleton and large deformations of tissue are adopted. In the first 10 s the rate of indentation amounts 1 mm/s and then the indenter stops. The total time of simulation is equal to 60 s. The area of contact of the indenter with tissue amounts 1 cm². Time dependence of indentation force, distributions and evolution of total stress and pore pressure being result of fluid outflow from the loaded domain are analysed for different values of permeability and stiffness. The obtained results approximate mechanical response of the lymphedematous tissue tested in vivo.

Keywords: computational modelling, geometric nonlinearity, lymphedema, mathematical model, soft human tissue

1. Introduction

The indentation test is frequently used by clinicians as an objective tool for diagnosis of edema and its evolution observed as the result of compression therapy. Due to the fact that soft biological tissues sustain large deformations, their behaviour is highly non-linear. Up till now there have been published a number of papers concerning mechanical response of the tissue subjected to the load of indenter on the basis of in vivo experiments [3,4,5]. In the literature there are several numerical models of skin and subcutaneous tissue in the indentation test which are done by finite element method, within the range of finite deformations (hyperelastic material models) [2,3,4]. However none of them consider lymphedematous tissue in the context of two-phase material, regarding geometrical and material nonlinearity. This work is based on mathematical model of fluid saturated porous medium taking into account large deformations of skeleton, proposed by Borja et al. [1]. The simplified geometry of tissue and cylindrical indenter (Fig. 1) is modeled by using finite element method implemented in COMSOL environment. The cylindrical coordinate system with axis \( z \) as a symmetry axis is adopted.

![Figure 1: Assumed geometry of the model](image)

The radius and height of the considered domain of subcutaneous tissue amounts 3 cm. The indenter’s radius has 5.5 mm and its height equals to 1 cm. The displacement of the indenter is a function of time: in the first 10 s its indentation rate amounts 1 mm/s and then the indenter stops. Control point PK, 5 mm under the indenter, is introduced to illustrate changes of stress and pore pressure in time. Arrows in Fig. 1 indicate the direction of pore fluid outflow.

2. Model of tissue

2.1. Governing equations

a) For fluid phase the continuity equation and Darcy’s law are assumed:

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \nabla \cdot (\rho \mathbf{u}) = Q_m, \quad \mathbf{u} = -\frac{k}{\eta} \nabla p
\]  

where \( \rho \) denotes the density of fluid, \( \varepsilon \) is the porosity and \( Q_m \) denotes mass source term; the discharge velocity \( \mathbf{u} \) is determined by pressure gradient \( \nabla p \), dynamic viscosity of the fluid \( \eta \) and permeability of the porous medium \( k \):

b) The solid skeleton is described by mechanical equilibrium equation, effective stress (the average stress carried by the solid skeleton) law, and hyperelastic (Neo-Hookean) material model:

\[
\nabla \cdot \sigma = 0, \quad \sigma = \bar{\sigma} - q \phi I
\]

\[
W_e = \frac{1}{2} \mu (I_1 - 3) - \mu (\ln(J_0) - 1) - \frac{1}{2} \lambda (\ln(J_0))^2
\]  

where \( \sigma \) and \( \bar{\sigma} \) denote the total and effective stress tensors, \( W_e \) is the elastic strain energy density that is the function of the elastic strain state, \( I_1 \) is the first deviatoric strain invariant, \( \alpha \) is the parameter of mechanical coupling, \( \mu \) and \( \lambda \) are Lame parameters (next expressed by Young’s modulus \( E \) and...
Poisson’s ratio \( \nu : \mu = G = \frac{E}{2(1+\nu)}, \quad \lambda = \frac{E\cdot \nu}{(1+\nu)(1-2\nu)}. \)

The elastic volume ratio \( J_{el} \) is defined by \( J_{el} = \det(F) \), where \( F = (I + \nabla u) \) is the deformation gradient.

2.2. Simulation parameters

The simulations are performed assuming the set of material parameters given in Tab. 1. The stiffness (Young modulus) and permeability of tissues were changed to analyse their role for mechanical response of tissue and pore fluid pressure dissipation related to outflow of interstitial fluid.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
<td>Volumetric coupling</td>
<td>1.000</td>
<td>-</td>
</tr>
<tr>
<td>( \rho )</td>
<td>Liquid density</td>
<td>1000.000</td>
<td>kg/m³</td>
</tr>
<tr>
<td>( \nu )</td>
<td>Poisson’s ratio</td>
<td>0.330</td>
<td>-</td>
</tr>
<tr>
<td>( E )</td>
<td>Young modulus</td>
<td>2, 2.5, 3·10⁴</td>
<td>Pa</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>Porosity</td>
<td>0.050</td>
<td>-</td>
</tr>
<tr>
<td>( k )</td>
<td>Permeability</td>
<td>0.5, 1.5, 4.5·10⁻¹³</td>
<td>m²</td>
</tr>
<tr>
<td>( \eta )</td>
<td>Dynamic viscosity</td>
<td>0.001</td>
<td>Pa·s</td>
</tr>
</tbody>
</table>

3. Results of simulations

Numerical calculations were done within the range of finite deformations. The displacement of the indenter was applied incrementally. At each time step the non-linear finite elements equations were solved using a Newton–Raphson method. Readings of mechanical response were taken at control point PK, placed 5 mm under the indenter.

![Figure 2: Time dependence of indentation force evaluated at the level of control point PK, for different stiffness values](image)

![Figure 3: Time dependence of stress at control point PK for different stiffness values](image)

![Figure 4: Time dependence of pore pressure at control point PK for different permeability values](image)

4. Conclusions

The obtained results showed significant influence of stiffness and permeability on mechanical response of lymphedematous tissue in the indentation test. For this reason the indentation test (tonometry) can be considered a meaningful diagnostic tool from which e.g. a maximum force implies how stiff is the tissue or taking a measure of relaxation of the tissue response the permeability can be evaluated. Next, the parameters may be used to optimise therapeutic methods.

References


Identification of mechanisms of attenuation and dispersion of ultrasonic waves in cancellous bone - theory and experiment

Michał Pakula
Institute of Mechanics and Computer Science, Kazimierz Wielki University in Bydgoszcz
Chodkiewicz 30, 85-064 Bydgoszcz, Poland
e-mail: michalp@kzu.edu.pl

Abstract

The paper presents theoretical and experimental issues related to application of Quantitative Ultrasound for the assessment of cancellous bone quality and prediction of bone fractures. Commonly used for modeling of ultrasonic wave propagation in cancellous bone, the Biot’s theory will be discussed in context of its potential applicability for theoretical prediction of wave parameters: phase velocity and attenuation coefficient as functions of frequency. The analysis of the model is focused on the absorption and scattering mechanisms responsible for attenuation of ultrasonic waves in cancellous bone. The suitability of the model is discussed by the comparison of results of sensitivity analysis of the model with experimental ultrasonic data obtained for cancellous bones filled with different fluids.

Keywords: wave attenuation, wave dispersion, cancellous bone, quantitative ultrasound, osteoporosis

1. Introduction

In the last two decades many studies examined relevance of skeletal quantitative ultrasound in context of early detection of osteoporosis [4, 5, 6, 7, 8, 10, 11]. Commonly used for modeling ultrasonic propagation in cancellous bone the macroscopic Biot’s theory will be discussed in context of its potential applicability for prediction of wave parameters: phase velocity and attenuation coefficient as functions of frequency. Considering trabecular bone as a fluid saturated porous material, the wave attenuation may result from: (i) intrinsic absorption in the fluid and solid phase, (ii) friction at the fluid-solid interface as well as (iii) wave scattering by inhomogeneities (pores/trabeculae). The suitability of the developed model is verified by comparison of results of sensitivity analysis of the model with experimental ultrasonic results obtained ex vivo for cancellous bones filled with different fluids. The aim of the paper is to examine the contribution of the properties of the solid and pore fluid to velocity dispersion and attenuation mechanisms in cancellous bone and verify the suitability of the developed Biot theory for modeling ultrasonic wave propagation in the material.

2. Modelling

Trabecular bone at the macroscopic level is a two-phase, anisotropic material composed of solid rod-like or plate-like skeleton filled in vivo with viscous fluid-like marrow. As theoretical basis for modeling elastic wave propagation in cancellous bone, mostly two-phase theory of dynamics of fluid-saturated porous materials proposed by Biot [1, 2] and extended by the other authors [9] is used. Accordingly to the Biot [1], the linearized two-phase model of wave propagation in the fluid saturated porous medium can be based on the two equations of linear momentum for the solid and fluid phases:

\[
\begin{align*}
\rho^s \frac{\partial \mathbf{T}^s}{\partial t} - \nabla \cdot \mathbf{T}^s &= \mathbf{R}^s \\
\rho^f \frac{\partial \mathbf{T}^f}{\partial t} - \nabla \cdot \mathbf{T}^f &= \mathbf{R}^f,
\end{align*}
\]

where \(\rho^s\) and \(\rho^f\) are the macroscopic mass density, \(\mathbf{T}^s\) and \(\mathbf{T}^f\) denote stress tensors while \(R^s\) and \(R^f\) stand for interaction force of linear momentum exchange of solid (s) and fluid (f) phase. For an isotropic porous material fully saturated, with an elastic skeleton, Biot postulated the relationships for stress and strain tensors in the form that can be found, elsewhere [1]. The elasticity constants can be related to physically well defined and measurable parameters of the porous medium, i.e. the porosity (\(\phi\)), the bulk modulus of the solid material \((K_s)\) and fluid \((K_f)\) and the bulk modulus of the drained skeleton \((K_0)\) [2]. The form of interaction forces proposed by Biot, include viscous and dynamic couplings, proportional to relative velocities and relative accelerations of phases [1].

Currently no consensus has been reached about the relevance of Biot’s theory for modeling of wave propagation in cancellous bone. A reasonable agreement between Biot’s theory and experiments has consistently been obtained when the phase velocity was considered [6, 10, 11]. However, in the case of attenuation, significant discrepancies were observed between predictions reported by different author [9, 11]. One of possible reason explaining poor agreement for attenuation coefficient between experiment and theory may be the fact, that in most of reported studies authors do not include absorption in the skeleton of the cancellous bone and wave scattering. Commonly the Biot’s model is used assuming elastic properties of the skeleton [6, 11]. Recently published experimental results show that two major mechanism of attenuation postulated in “elastic” Biot’s model i.e. viscous properties of the fluid and the friction at the fluid-solid interface, play a minor role in total attenuation and that the wave energy is presumably reduced by the properties of the skeleton [8, 10]. Following the experimental observations, it was proposed to include the visco-elastic properties of skeleton and scattering effects in the model, by complex elasticity constants [2].

3. Ultrasonic Experiments

The experiments were performed for 26 human cancellous bone specimens obtained from the distal end of human femurs.
After the initial measurements performed on marrow-saturated specimens, all the specimens were defatted, water and alcohol saturated and remeasured using the same method. Experiments were done in immersion using three pairs of unfocused ultrasonic transducers with center frequencies of 0.5, 1, and 2 MHz [8]. In Fig. 1 and Fig. 2 comparison of experimental ultrasonic results and theoretical predictions of Biot model are presented for phase velocity and attenuation coefficient, respectively.

Considering the phase velocity (Fig. 1) it is visible that elastic properties of the fluid determine the values of the velocity in cancellous bone. The values of the velocities are practically constant within the whole frequency range. Attenuation coefficient (Fig. 2) is almost linear function of frequency. Moreover, surprisingly it is practically independent on the viscous properties of fluid filling specimens.

4. Summary and Conclusions

The purpose of the paper was to examine experimentally and theoretically the contribution of the properties of the solid and pore fluid to velocity dispersion and attenuation mechanisms in cancellous bone. It was found experimentally, that the elastic properties of the fluid are main determinants of the phase velocity. The attenuation remains nearly linearly frequency dependent in a broad frequency range. It was shown that the fluid properties does not influence the attenuation coefficient of cancellous bone between 0.35 and 2.5 MHz. Our study clearly demonstrated that the viscosity of the saturating fluid did not influence the amplitude of the attenuation coefficient neither its frequency dependence. We therefore hypothesize that the source of wave attenuation can be associated with viscoelastic absorption in the solid phase or/and with scattering by the solid trabeculae.

Following experimental conclusions concerning absorption like mechanism of ultrasonic wave attenuation caused mostly by skeleton, the Biot’s model was developed including viscoelastic properties of the solid component and the scattering effects. A good quantitative and qualitative agreement was found between wave parameters (phase velocity and attenuation coefficient as a function of frequency) measured experimentally and predicted by developed Biot’s model.

Based on obtained results one can conclude that the theoretical predictions of commonly used for modeling elastic wave propagation in cancellous bone Biot theory for wave propagation in a fluid saturated porous medium may be a useful tool for modeling ultrasound propagation the bone material.

References

Cancer ablation during RF hyperthermia using internal electrode

Marek Paruch

Institute of Computational Mechanics and Engineering, Silesian University of Technology
Konarskiego 18A, 44-100 Gliwice, Poland

Abstract

The paper presents numerical modeling of RF (radiofrequency) hyperthermia caused by the introduction of internal electrode to the tumor. The main purpose of the publication is to analyze the relationship between electrode voltage and the duration of hyperthermia treatment necessary to achieve adequate temperature in the tumor tissue. Mathematical modeling based on the coupling of two problems: electrical - to generate additional heat and thermal - to estimate the change and rise of the temperature in the tumor. The distribution of electric potential in the tissue is described by the Laplace equation, while the temperature field is described by the Pennes equation. At the stage of numerical simulation the boundary element method (BEM) is used. In the final part of the paper the results of numerical computations are shown.

Keywords: bioheat transfer, boundary element method, cancer ablation, internal electrode, RF hyperthermia

1. Introduction

Artificial hyperthermia is a method of treating cancer, therein the cancer tissues are subjected to external [5] or internal [4] impacts to high temperatures - up to 45°C. Hyperthermia, or overheating of the tumor, becomes one of the esteemed methods of dealing with cancer. It is a kind of radiotherapy, which can significantly increase the effectiveness of cancer treatment. During hyperthermia, the tissues are subjected to the temperatures of 40 to 45°C, because temperatures above 42°C begins the process of necrosis of living cells and the temperatures above 45°C are known as thermoablative [2]. Prolonged exposure at such high temperature destroys the cells via coagulation necrosis. Research have shown that high temperatures can damage and kill cancer cells, usually accompanied by a low risk of damage to healthy tissue [4].

2. Mathematical modelling

RF hyperthermia (RFH) represents coupled electro-thermal problems, and is applicable to use of electromagnetic energy to heat a specific area. Therefore it is important to correctly describe how the electromagnetic field interacts with biological tissue. In the full Maxwell’s equations, electric and magnetic fields are coupled, so it requires complex numerical calculations. Because the RFH uses low frequencies, in order to determine the intensity of the electric field the simplification known as the quasi-static approach can be taken into account.

Using the quasi-static formulation, the electric field intensity $E$ (V/m) inside the tissue for 2D problem can be calculated as follows [1, 5]

$$E(X) = -\nabla \varphi(X) = [-\varphi_x(X) / \partial x, \varphi_y(X) / \partial y]$$

where $e=1$ denotes the healthy tissue and tumor, respectively, $X=[x_1, x_2]$ and $\varphi$, [V] is an electric potential, while the heat generation $Q^\mathcal{E}(X)$ due to the electromagnetic heating is defined as follows

$$Q^\mathcal{E}(X) = \frac{\sigma_a |E(X)|^2}{2} = \frac{\sigma_a}{2} \sum_i \left( \frac{\partial \varphi_i(X)}{\partial x} \right)^2$$

where $\sigma_a$ [S/m] is an electrical conductivity.

The electric potential $\varphi(X)$ inside the healthy tissue $\Omega_1$ and cancer $\Omega_2$ (c.f. Figure 1) is described by the system of Laplace equations

$$X \in \Omega_i : \nabla [\varepsilon_e(X) \nabla \varphi(X)] = 0$$

where $\varepsilon_e$ [C/(Nm²)] is a dielectric permittivity of tissue.

For heat transfer process in biological tissue the Pennes model has been proposed [6]

$$c_e \rho_e \frac{\partial T_e(X)}{\partial t} = \lambda_e \nabla^2 T_e(X) + k_e \left(T_B - T_e(X)\right) + Q_{met} + Q^\mathcal{E}$$

where $t$ denotes time, $\rho_e$ [kg/m³] is the density, $c_e$ [J/(kgK)] is the specific heat, $\lambda_e$ [W/(mK)] is the thermal conductivity, $T_e$ [K] is the temperature, $k_e = G_{be} c_B$ [W/(m³K)] is the perfusion rate $(G_{be} [1/s]$ is the perfusion coefficient, $c_B$ [J/(m³K)] is the volumetric specific heat of blood), $T_B$ is the supplying arterial blood temperature and $Q_{met}$ [W/m³] is the metabolic heat source.

Differential equations which describe the electric (c.f. eq. (3)) and temperature (c.f. eq. (4)) fields are supplemented by appropriate boundary-initial conditions. Constant voltage boundary condition $\varphi=U$ ($U[V]$ is an electric potential of the electrode) is applied on the active part of the electrode (tip length: 1.5 cm), whereas the boundaries $\Gamma_1$, $\Gamma_2$ and $\Gamma_3$ are considered as the ground ($U=0$). On the remaining boundaries electric insulation is assumed. In the case of temperature field, boundaries $\Gamma_1$-$\Gamma_3$, the body core temperature $T_B=37°C$ is assumed, while on the electrode surface the thermal insulation is applied. On the contact surface between healthy tissue and tumor the ideal electric and thermal contacts were assumed. Equation (4) is also supplemented by the initial condition $T_e(X)=T_{init}$. 
3. Boundary element method

In order to solve the equations describing the potential of electric field and the temperature field in the considered domains the boundary element method has been applied [3].

The boundary integral equations corresponding to the equations (3) can be expressed as follows

\[ B_e(\xi, \eta) \phi_e(\xi, \eta, X) + \int_B \psi_e(X) \phi_e(\xi, \eta, X) d\Gamma = \int_B \phi_e(X) \psi_e(\xi, \eta, X) d\Gamma, \quad e = 1, 2 \]

while for temperature field (c.f. eq.(4))

\[ B_e(\xi, \eta) T_e(\xi, \eta, t') + \frac{1}{c} \int_B T_e(\xi, \eta, X, t', t) q(X, t) d\Gamma d\Omega = \]

\[ \frac{1}{c} \int_B \int_0^t q_e(\xi, \eta, X, t') q(X, t') d\Omega d\Gamma + \]

\[ \int_B T_e(\xi, \eta, X, t', t') T_e(\xi, \eta, X, t') d\Omega + \]

\[ \frac{1}{c} \int_B \int_B Q_e(X, t) T_e(\xi, \eta, X, t') d\Omega d\Gamma, \quad e = 1, 2 \]

where \( Q_e = Q_{met} + Q_{ph} \). The functions \( \phi_e(\cdot) \) and \( T_e(\cdot) \) are the fundamental solutions. The above boundary-integral equations are solved using the method described in [3].

4. Results of computations

The 2D domain of dimensions 0.16x0.12 [m] has been considered. The tumor region which is located at the center of healthy tissue (c.f. Figure 1) is approximated by circle (radius: 0.025 [m]). The electrical and thermophysical parameters are collected in Tables 1 and 2, respectively.

The initial temperature \( T_{init} = 37^\circ C \), time step \( \Delta t = 5s \) and maximum analysis time is assumed as 1h.

Table 3 contains the results of numerical calculations for various values of voltage on the electrode together with the information about the time in the center of tumor the temperature increase to the value 45°C was observed.

<table>
<thead>
<tr>
<th>U [V]</th>
<th>Exposure time t [s]</th>
<th>Temp. at the control node (c.f. Figure 1) [°C]</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>3600</td>
<td>40.17</td>
</tr>
<tr>
<td>15</td>
<td>3600</td>
<td>43.56</td>
</tr>
<tr>
<td>17</td>
<td>1500</td>
<td>45.01</td>
</tr>
<tr>
<td>20</td>
<td>340</td>
<td>45.01</td>
</tr>
<tr>
<td>25</td>
<td>130</td>
<td>45.09</td>
</tr>
</tbody>
</table>

In Figure 2 the temperature field obtained after the action of the internal electrode (\( U=17[V] \), \( t=1500s \)) is shown.

Table 1: The electrical parameters (c.f. eqs. (2), (3))

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Healthy tissue</th>
<th>Tumor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dielectric permittivity</td>
<td>8.85x10^{-10}</td>
<td>1.3275x10^{-9}</td>
</tr>
<tr>
<td>Electrical conductivity</td>
<td>0.2</td>
<td>0.3</td>
</tr>
</tbody>
</table>

Table 2: The thermophysical parameters (c.f. eq. (4))

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Healthy tissue</th>
<th>Tumor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal conductivity</td>
<td>0.5</td>
<td>0.75</td>
</tr>
<tr>
<td>Perfusion rate</td>
<td>1998.1</td>
<td>7992.4</td>
</tr>
<tr>
<td>Metabolic heat source</td>
<td>420</td>
<td>4200</td>
</tr>
<tr>
<td>Arterial blood temperature</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Density</td>
<td>1090</td>
<td></td>
</tr>
<tr>
<td>Specific heat</td>
<td>3421</td>
<td></td>
</tr>
</tbody>
</table>

Table 3: Results of computations

References


Remarks on least-squares finite element formulations for hyperelasticity

Jörg Schröder¹, Karl Steeger², Alexander Schwarz³ *

¹,²,³ Institut für Mechanik, Universität Duisburg-Essen
Universitätstraße 15, 45141 Essen, Germany
e-mail: j.schroeder@uni-due.de ¹, karl.steeger@uni-due.de ², alexander.schwarz@uni-due.de ³

Abstract

In the present contribution least-squares finite element method (LSFEM) formulations in the field of hyperelasticity are taken under consideration. Basis for the element is a div-grad first order system consisting of the equilibrium condition and the constitutive equation both written in a residual form. In order to define the stress response of the material, different hyperelastic polyconvex free energy functions will be presented. The sum of squared $L_2$ norms of the residuals leads to the basic functional which has to be minimized. Therefore the first variations with respect to both free variables have to be zero. The solution of this nonlinear system of equations can then be found by applying Newton’s Method. For the approximation of the displacement and the stress field we use independent interpolations. In order to show the performance of the proposed method different boundary value problems will be analyzed and remarks concerning bifurcation points will be adressed.

Keywords: least-squares, mixed finite elements, hyperelasticity

1. Introduction

In this contribution we focus on finite elasticity and assume the existence of a free energy function $\psi$. For the discretization and solution of the governing differential equations a least-squares finite element method (LSFEM) is applied. In order to derive the constitutive relation hyperelastic free energy functions of Neo-Hookean and Mooney-Rivlin type are used. The considered functional is the basis for a displacement-stress formulation and solution of the governing differential equations a least-squares functional which has to be minimized. Therefore, we need the first variations to be zero

$$\delta_u \mathcal{F} = \sum_{i=1}^{3} \int_B w_i \delta_u \mathcal{R}_i \cdot \mathcal{R}_i \, dV = 0,$$

$$\delta_P \mathcal{F} = \sum_{i=1}^{3} \int_B w_i \delta_P \mathcal{R}_i \cdot \mathcal{R}_i \, dV = 0$$

and the second variations for the application of a Newton scheme to find the root

$$\Delta_u \delta_u \mathcal{F} = \sum_{i=1}^{3} \int_B w_i (\Delta_u \delta_u \mathcal{R}_i \cdot \mathcal{R}_i + \delta_u \mathcal{R}_i \cdot \Delta_u \mathcal{R}_i) \, dV,$$

$$\Delta_P \delta_u \mathcal{F} = \sum_{i=1}^{3} \int_B w_i (\Delta_P \delta_u \mathcal{R}_i \cdot \mathcal{R}_i + \delta_u \mathcal{R}_i \cdot \Delta_P \mathcal{R}_i) \, dV,$$

$$\Delta_u \delta_P \mathcal{F} = \sum_{i=1}^{3} \int_B w_i (\Delta_u \delta_P \mathcal{R}_i \cdot \mathcal{R}_i + \delta_P \mathcal{R}_i \cdot \Delta_u \mathcal{R}_i) \, dV,$$

$$\Delta_P \delta_P \mathcal{F} = \sum_{i=1}^{3} \int_B w_i (\Delta_P \delta_P \mathcal{R}_i \cdot \mathcal{R}_i + \delta_P \mathcal{R}_i \cdot \Delta_P \mathcal{R}_i) \, dV.$$  (5)

with

$$\delta_u \mathcal{R}_1 = 0, \quad \delta_P \mathcal{R}_1 = \text{Div} \delta P,$$

$$\delta_u \mathcal{R}_2 = -\rho_0 \partial^2 \psi(C) \delta F, \quad \delta_P \mathcal{R}_2 = \delta P,$$

$$\delta_u \mathcal{R}_3 = \delta F^{-1} \delta P - (\delta F^{-1} \delta P)^T,$$

$$\delta_P \mathcal{R}_3 = \delta F^{-1} \delta P - (\delta F^{-1} \delta P)^T,$$

$$\Delta_u \mathcal{R}_1 = 0, \quad \Delta_P \mathcal{R}_1 = \text{Div} \Delta P, \ldots$$  (6)

2. Theoretical framework

2.1. General geometrically nonlinear least-squares approach

For the construction of a least-squares functional, the square of a $L^2$-norm i.e.

$$||a(x)||_{L^2(B)} = \int_B |a(x)|^2 \, dV,$$  (1)

where $B$ denotes the body of interest, is applied on a first-order system of differential equations written in residual form as

$$\mathcal{R}_1 = \text{Div} P + f \to \text{balance of momentum},$$

$$\mathcal{R}_2 = P - \rho_0 \partial \psi(C) \to \text{constitutive relation},$$

$$\mathcal{R}_3 = F^{-1} P - (F^{-1} P)^T \to \text{stress symmetry}.$$  (2)

For the details of the theoretical approach of the LSFEM and the rules for the construction of least-squares functionals the reader is referred to [2]. With the first Piola-Kirchhoff stress tensor $P$, the body force $f$, the density $\rho_0$, the deformation gradient $F$, the right Cauchy-Green tensor $C = F^T F$, the weights $w_i$ and the free energy $\psi$ we obtain the weighted least-squares functional

$$\mathcal{F}(P, u) = \sum_{i=1}^{3} \int_B w_i |\mathcal{R}_i|^2 \, dV$$  (3)

which has to be minimized. Therefore, we need the first variations to be zero

$$\delta_u \mathcal{F} = \sum_{i=1}^{3} \int_B w_i \delta_u \mathcal{R}_i \cdot \mathcal{R}_i \, dV = 0,$$

$$\delta_P \mathcal{F} = \sum_{i=1}^{3} \int_B w_i \delta_P \mathcal{R}_i \cdot \mathcal{R}_i \, dV = 0$$

and the second variations for the application of a Newton scheme to find the root

$$\Delta_u \delta_u \mathcal{F} = \sum_{i=1}^{3} \int_B w_i (\Delta_u \delta_u \mathcal{R}_i \cdot \mathcal{R}_i + \delta_u \mathcal{R}_i \cdot \Delta_u \mathcal{R}_i) \, dV,$$

$$\Delta_P \delta_u \mathcal{F} = \sum_{i=1}^{3} \int_B w_i (\Delta_P \delta_u \mathcal{R}_i \cdot \mathcal{R}_i + \delta_u \mathcal{R}_i \cdot \Delta_P \mathcal{R}_i) \, dV,$$

$$\Delta_u \delta_P \mathcal{F} = \sum_{i=1}^{3} \int_B w_i (\Delta_u \delta_P \mathcal{R}_i \cdot \mathcal{R}_i + \delta_P \mathcal{R}_i \cdot \Delta_u \mathcal{R}_i) \, dV,$$

$$\Delta_P \delta_P \mathcal{F} = \sum_{i=1}^{3} \int_B w_i (\Delta_P \delta_P \mathcal{R}_i \cdot \mathcal{R}_i + \delta_P \mathcal{R}_i \cdot \Delta_P \mathcal{R}_i) \, dV.$$  (5)

with

$$\delta_u \mathcal{R}_1 = 0, \quad \delta_P \mathcal{R}_1 = \text{Div} \delta P,$$

$$\delta_u \mathcal{R}_2 = -\rho_0 \partial^2 \psi(C) \delta F, \quad \delta_P \mathcal{R}_2 = \delta P,$$

$$\delta_u \mathcal{R}_3 = \delta F^{-1} \delta P - (\delta F^{-1} \delta P)^T,$$

$$\delta_P \mathcal{R}_3 = \delta F^{-1} \delta P - (\delta F^{-1} \delta P)^T,$$

$$\Delta_u \mathcal{R}_1 = 0, \quad \Delta_P \mathcal{R}_1 = \text{Div} \Delta P, \ldots$$  (6)

* DFG Grant SCHR 570/14-1: Gemischte Least-Squares Finite Elemente für geometrisch nichtlineare Probleme der Festkörpermechanik
and
\[ \Delta u \delta_R R_1 = \Delta p \delta_R R_1 = \Delta u \delta_R R_4 = \Delta p \delta_R R_1 = 0, \]
\[ \Delta u \delta_R R_2 = - \partial_F (\partial_F \psi(C) \partial F) \Delta F, \]
\[ \Delta p \delta_R R_2 = \Delta p \delta_R R_2 = 0, \]
\[ \Delta u \delta_R R_3 = \Delta p \delta_R R_3 = 0, \]
\[ \Delta p \delta_R R_3 = \Delta F^{-1} \delta F - (\Delta F^{-1} \delta F)^T, \]
\[ \Delta u \delta_R R_3 = \Delta F^{-1} \delta F - (\Delta F^{-1} \delta F)^T. \] (7)

2.2. Interpolation

The interpolation of the unknowns is performed by the approximation spaces \( W^{1,p}(\mathcal{B}) \) and \( W^p(\text{div}, \mathcal{B}) \) with suitable \( p \), compare also [3]. For \( W^{1,p}(\mathcal{B}) \) (respectively \( W^p(\text{div}, \mathcal{B}) \)) that means, that the functions and their derivative (gradient, divergence) must be in \( L^p(\mathcal{B}) \)
\[ W^{1,p}(\mathcal{B}) = \{ \mathbf{a} \in L^p(\mathcal{B}) : \nabla \mathbf{a} \in L^p(\mathcal{B}) \} \] (8)
and
\[ ||\mathbf{a}||_{L^p(\mathcal{B})} = \left( \int_{\mathcal{B}} |\nabla \mathbf{a}|^p dV \right)^{\frac{1}{p}}. \] (9)

Using polynomial interpolation for the approximation of \( W^{1,p}(\mathcal{B}) \) and vector-valued Raviart-Thomas functions for the conforming discretization of the Sobolev space \( W^p(\text{div}, \mathcal{B}) \) we obtain, depending on the choice of the interpolation space for each free variable, elements of the types \( RT_m \), \( P_k \) (stresses in \( W^p(\text{div}, \mathcal{B}) \) and displacements in \( W^{1,p}(\mathcal{B}) \)), compare also [4] with \( m \) and \( k \) denoting the respective interpolation order.

2.3. Material model

For the description of the stress response of the materials, we use the derivative of a free energy function based on the invariants of the right Cauchy-Green deformation tensor \( C \). The invariants are given as
\[ I_1 = \text{tr} C, \]
\[ I_2 = \text{tr} \left( \cot C \right) = \frac{1}{2} \left[ (\text{tr} C)^2 - (\text{tr} C^2) \right] \]
\[ I_3 = \det C = (\text{det} F)^2. \] (11)

We consider a free energy function of Neo-Hookean type given as
\[ \psi^{NH} = \frac{\lambda}{4} (I_3 - 1) - \left( \frac{\lambda}{2} + \mu \right) \ln(\sqrt{I_3}) + \frac{\mu}{2} (I_1 - 3) \] (12)
and of Mooney-Rivlin type with \( \mu = \mu_1 + \mu_2 \) as
\[ \psi^{MR} = \frac{\lambda}{4} (I_3 - 1) - \left( \frac{\lambda}{2} + \mu_1 + 2 \mu_2 \right) \ln(\sqrt{I_3}) + \frac{\mu_1}{2} (I_1 - 3) - \frac{\mu_2}{2} (I_2 - 3), \] (13)
with the material parameters \( \lambda, \mu, \mu_1, \mu_2 \). The first Piola-Kirchhoff stress tensor is now given as the derivative of the free energy with respect to the deformation gradient \( F \). We obtain for (12) the first Piola-Kirchhoff stress tensor as
\[ \mathbf{P} = \frac{\lambda}{2} (I_3 - 1) \mathbf{F}^{-T} + \mu \left( \mathbf{F} - \mathbf{F}^{-T} \right) \] (14)
and for (13) as
\[ \mathbf{P} = \left( \frac{\lambda}{2} (I_3 - 1) - \mu_1 + 2 \mu_2 \right) \mathbf{F}^{-T} + \mu_1 \mathbf{F} - \mu_2 F (I_1 I - C), \] (15)
compare [5].

3. Numerical example

As an introducing example we want to show a three-dimensional uni-axial tension test where we consider \( RT_0 \) elements for both materials. The geometry is given by a cube with an edge length of 3, which is statically determined. All stress-free faces are bounded by essential boundary conditions for the respective nodal degrees of freedom for the stresses. The material parameters used are
\[ \lambda = 172.8, \mu = 47.1 \quad \text{and} \quad \mu_1 = 57.1. \] (16)

In \( x \)-direction a load \( \mathbf{P} = (P_{xx}, 0)^T \) is applied resulting in an elongation of the cube. To show the different material behavior for Neo-Hooke and Mooney-Rivlin and verify the formulation the load is plotted over the deflection of a corner node, see Fig. 1, and compared with a reference solution calculated by a standard Galerkin finite element.

![Figure 1: Stress-displacement diagram for both materials calculated by a uniaxial tension test](image)

References


Numerical simulation of the traction process in the treatment for Robin Sequence

Jakub J. Słowiński¹, Aleksandra Czarnecka²

¹,² Faculty of Mechanical Engineering, Wrocław University of Technology
Smoluchowskiego 25, 50-370 Wrocław, Poland
e-mail: jakub.slowinski@pwr.edu.pl

Abstract

The Robin Sequence (RS) is a developmental malformation characterized by a triad of symptoms: cleft palate, micrognathia and glossoptosis. In the process of treating the RS the first non-invasive methods are considered (prone positioning, short-term intubation), while in more severe cases surgical interventions (traction, osteodistraction) are preferred. The aim of the study was to develop a model and a simulation of an increase in the length of the jaw bone under the influence of the method of traction therapy.

The developed procedure simulates a 20-week traction using volume and plane models. A finite volume model of infant mandible was developed based on a CT scan of an adult mandibular bone and cephalometric documentation of a newborn suffering from RS while a plane model was based on the cross-section of a 3d model. In each of the analysed variants an increase in the rate of growth compared to the unmodulated growth was reported.

Keywords: Robin sequence, FEM, bone growth simulation, traction

1. Introduction

The Robin Sequence (RS) is a developmental malformation characterized by a triad of symptoms: cleft palate, micrognathia and glossoptosis. The most characteristic feature of this syndrome is micrognathia manifested by significantly reduced mandibular bone dimensions in all directions [1]. In the process of treating the RS the first non-invasive methods are considered (prone positioning, short-term intubation), while in more severe cases surgical interventions (traction, osteodistraction) are preferred [Poets]. The aim of this study was to develop a model and a simulation of an increase in the length of the jaw bone under the influence of the method of traction therapy.

2. Materials and methods

The developed procedure simulates a 20-week period traction. It was assumed that constant (unmodulated) growth of the bone in the area of the mandibular branch is equal to 0.35 mm per week [1]. Using the 3d model (Fig. 1A-D) of the mandibular bone loaded by forces typical for the traction process a displacement distribution was obtained. Then, based on the cross-section of the 3d model a plane model was created and the previously calculated displacements were assigned. A finite volume model of an infant mandible was developed based on the CT scan of an adult mandibular bone and cephalometric documentation of a newborn suffering from RS (Fig. 1). Discretization was carried out using a 20-node, high-order hexahedral element with 3 degrees of freedom in every node. The discretized model was divided into 3 parts (Fig. 2). The first part were condylar and coronoid processes filled with a cancellous bone tissue and covered with a cartilage. The second part was a mandibular branch, where the growing process was carried out – fully modelled as a cancellous bone tissue. The base of mandible constituted the last, third part in which it was filled with a cancellous bone tissue and covered with a compact bone tissue. All tissues were simulated as linear, isotropic materials. The material properties of all tissues (Table 1) were taken based on literature data [2,3]. The model was fixed in the area of temporomandibular joint and the coronoid process and then loading forces to selected nodes (2 N and 5 N) were assigned – Fig. 1E.

There were three variants of loading conditions – forces were applied from three different angles (10°, 20°, 30°) – Fig. 1E. Preliminary results from the volume model were used to apply the boundary conditions to the plane model which was used to simulate the mandibular bone growth process. The proposed algorithm, in accordance with statistical data, simulated mechanical modulation and lengthening of the mandibular branch. As a measure of growth the authors assumed the increase in the length of the mandible, defined as the distance between the anatomical points Articulare and Pogonion articular (ar-Pog), used in the cephalometric analysis.

The plane model was discretized using a 4-node quad element with 2 degrees of freedom at every node.

![Figure 1: Volume model of the mandible based on the CT scan of an adult mandible and cephalometric documentation of a newborn suffering from RS (A-D), a scheme of boundary conditions (E)](image-url)
Table 1: Material properties of modelled tissues [2][3]

<table>
<thead>
<tr>
<th>Tissue</th>
<th>Young modulus [MPa]</th>
<th>Poisson coefficient [-]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compact bone</td>
<td>14000</td>
<td>0.3</td>
</tr>
<tr>
<td>Cancellous bone</td>
<td>500</td>
<td>0.12</td>
</tr>
<tr>
<td>Cartilage</td>
<td>23.8</td>
<td>0.4</td>
</tr>
</tbody>
</table>

Figure 2: The plane model of the mandible (left) and the cross-section of the volume model were taken from boundary conditions for the plane model.

The procedure began the growth rate (G) calculation, which in the model means an increase in the mandibular branch length. This parameter was calculated on the basis proposed by the Stokes equation of the mechanical modulation of growth:

\[ G = G(1 + \beta M) \]

where \( G \) is non-modulated bone growth [mm/week], \( \beta \) is experimentally determined coefficient equal 1.71 MPa\(^{-1}\), \( M \), [MPa] is the mean value of Mises stress occurring in the area of growth.

3. Results

In each of the six analysed variants (2 forces, 3 angles) an increase in the rate of growth compared to the unmodulated growth (Fig. 3) was reported. The increase in mandibular length was linear. The highest lengthening was observed using a load of 5 N and the angle of force application equal to 10° – it amounted to 7.82 mm (Fig. 4). This means that in the most profitable case the mechanical modulation increase of the growth was 26.3% (2 forces, 3 angles).

the growth values also decreased. With the increase of the angle of force application, both the maximum stress values, as well as the size of the area where they occur, were reduced.

Figure 4: 3 stages of the traction process – first, middle and final, force 5 N, 10° – in this variant the maximum lengthening were carried out

References


Modelling and analysis of a contact problem with unilateral constraint

Mirecea Sofonea\(^1,2\), Mikael Barboteu\(^2\)

\(1,2\) Laboratoire de Mathématiques et Physique, Université de Perpignan Via Domitia
52 Avenue Paul Alduy, 66 860 Perpignan, France
\(e\)-mail: sofonea@univ-perp.fr \(^1\), barboteu@univ-perp.fr \(^2\)

Abstract

We consider a mathematical model describing a frictionless contact between a viscoplastic body and a foundation. The process is quasistatic and the contact is modeled with normal compliance and unilateral constraint. Moreover, the stiffness coefficient of the foundation is allowed to depend on the history of the contact process. We derive a variational formulation of the problem, then we state and prove its unique weak solvability and the dependence of the weak solution with respect to the data. The proofs are based on the results from the theory of history-dependent variational inequalities, various estimates and monotonicity arguments. Further, we present a numerical method for solving the problem and we implement a solution algorithm. We also report numerical simulation, which illustrates the mechanical behavior related to the contact condition.

Keywords: viscoplastic material, frictionless contact, normal compliance, unilateral constraint, history-dependent variational inequality, weak solution, continuous dependence, finite element scheme, numerical simulations.

1. Introduction

Phenomena of contact involving deformable bodies abound in industry and everyday life. Contact of braking pads with wheels, tires with roads, pistons with skirts are just a few simple examples. Common industrial processes such as metal forming or metal extrusion involve the evolution of the contact processes and, for these reasons, a considerable effort has been made in their modeling, mathematical analysis, and numerical simulations. Owing to their inherent complexity and unusual structure, contact phenomena lead to nonlinear and nonsmooth mathematical problems. The mathematical tools for their analysis, including proofs of the existence and possible uniqueness of the solution, were developed in a number of works, see for instance [4] and the references therein. A recent reference in the field is the special issue [2] which provides the state of the art review of the Mathematical Theory of Contact Mechanics and presents some of its applications in engineering.

It follows from the references above that the modelling of contact process is an important topic which, currently, is still under investigation. The contact boundary conditions represent one of the main ingredients in the construction of mathematical models of contact. A well-known example, used both in the engineering and the mathematical literature, is the famous Signorini contact condition, see [4] for details. The Signorini condition describes the contact with a perfectly rigid foundation, is expressed in terms of unilateral constraints and, therefore, it leads to highly nonlinear and nonsmooth mathematical problems. For this reason, the normal compliance contact condition was introduced in [3], and used in a large number of papers. It represents a regularization of the Signorini contact condition and it describes the contact with an elastic foundation. A more general contact condition, called the normal compliance condition with unilateral constraint, was introduced in [1]. It models the contact with an elastic-rigid foundation and it contains as particular cases both the Signorini contact condition and the normal compliance condition. It can be formulated as follows:

\[
\forall \nu \in \mathbb{R}^d, \quad \sigma_{\nu} + p(u_{\nu}) \leq 0, \quad (u_{\nu} - g)(\sigma_{\nu} + p(u_{\nu})) = 0. \quad (1)
\]

Here and below \(u_{\nu}\) and \(\sigma_{\nu}\) represent normal displacement and normal stress on the contact surface, respectively, and \(p\) is a given nonnegative function which vanishes for negative arguments. Condition (1) can be naturally interpreted as follows: the foundation is made of a hard material covered by a thin layer of a softer material with thickness \(g > 0\); the soft material has an elastic behavior, i.e., is deformable and allows penetration; the contact with this layer is modeled with normal compliance; the hard material is perfectly rigid and, therefore, it does not allow penetration; the contact with this material is modeled with the Signorini contact condition.

Our aim in the paper is to provide the analysis of a new model of contact which is based on a history-dependent version of the contact condition (1). This includes its unique weak solvability, the continuous dependence of the weak solution with respect to the problem data, and its numerical approach.

2. The model and main results

We consider a viscoplastic body that occupies a bounded domain \(\Omega\) with the boundary \(\partial \Omega = \Gamma\) partitioned into three disjoint measurable parts \(\Gamma_1, \Gamma_2\) and \(\Gamma_3\), such that \(\text{meas}(\Gamma_1) > 0\). Let \([0, T]\) be the time interval of interest with some \(T > 0\). The body is clamped on \(\Gamma_1 \times (0, T)\) and, therefore, the displacement field vanishes there. A volume force of density \(f_\Omega\) acts in \(\Omega \times (0, T)\), surface tractions of density \(f_\Sigma\) act on \(\Gamma_2 \times (0, T)\) and, finally, we assume that the body is in contact with a deformable foundation on \(\Gamma_3 \times (0, T)\). The contact is frictionless and we model it with normal compliance and unilateral constraint. The novelty here is that the stiffness coefficient in the normal compliance condition depends on the accumulation of penetration. Denote by \(S^d\) the space of second order symmetric tensors on \(\mathbb{R}^d\). Then, the classical formulation of the contact problem is the following.
Problem $P$. Find a displacement field $u : \Omega \times [0, T] \to \mathbb{R}^d$ and a stress field $\sigma : \Omega \times [0, T] \to \mathbb{S}^d$ such that

$$\dot{\sigma} = \mathcal{E} \varepsilon(u) + G(\sigma, \varepsilon(u)) \quad \text{in} \quad \Omega \times (0, T),$$

(2)

$$\text{Div} \sigma + \mathbf{f}_0 = 0 \quad \text{in} \quad \Omega \times (0, T),$$

(3)

$$u = 0 \quad \text{on} \quad \Gamma_1 \times (0, T),$$

(4)

$$\sigma \nu = f_2 \quad \text{on} \quad \Gamma_2 \times (0, T),$$

(5)

$$u_{\nu} \leq g, \quad \sigma_{\nu} + K_p(u_{\nu}) \leq 0, \quad (u_{\nu} - g)(\sigma_{\nu} + K_p(u_{\nu})) = 0 \quad \text{on} \quad \Gamma_3 \times (0, T),$$

(6)

$$\sigma_{\nu} = 0 \quad \text{on} \quad \Gamma_3 \times (0, T),$$

(7)

$$u(0) = u_0, \quad \sigma(0) = \sigma_0 \quad \text{in} \quad \Omega.$$  

(8)

Equation (2) represents the viscoplastic constitutive law of the material in which $\mathcal{E}$ and $G$ are given constitutive functions and the dot above represents the derivative with respect to the time. Equation (3) is the equilibrium equation and we use it here since the process is assumed quasistatic. Conditions (4) and (5) are the displacement and traction boundary conditions, respectively, in which $\nu$ represents the unit outward normal on $\Gamma$. Condition (6) represents a contact condition of the form (1) in which $K$ is an unknown function defined by equality:

$$K(t) = k \left( \int_0^t u^+_{\nu}(s) \, ds \right),$$

(9)

$u^+_{\nu}$ being the positive part of $u_{\nu}$ and $k$ a given positive function. Thus, the function $K$ represents the stiffness coefficient of the foundation and depends on the accumulation of penetration. As the cycles of contact and separation proceed, $K$ may increase or decrease, describing in this way the hardening or the softening of the foundation. Condition (7) shows that the tangential stress on the contact surface, denoted $\sigma_{\nu}$, vanishes. We use it here since we assume that the contact process is frictionless. Finally, (8) represents the initial conditions, $u_0$ and $\sigma_0$ being the initial displacement and the initial stress field, respectively.

In the study of $P$ we provide: a variational formulation which is in a form of a system coupling an implicit integral equation for the stress field and a history-dependent variational inequality for the displacement field; an existence and uniqueness result for the weak solution; a continuous dependence result of the weak solution with respect to the functions $p$ and $k$; a numerical method for solving the problem together with a solution algorithm; numerical simulations which validate our theoretical result.

3. Numerical simulations

Consider now a representative academic example which concerns the compression of a ball against a foundation. Due to the symmetry of the problem, we only need to consider the physical setup depicted in Figure 1. The domain $\Omega$ represents the cross-section of a three-dimensional ball subjected to the action of tractions in such a way that a plane stress hypothesis is assumed. The horizontal component of the displacement field vanishes on $\Gamma_1$ and vertical tractions act on $\Gamma_2$. The ball is in frictionless contact with an obstacle on the part $\Gamma_3$ of its boundary. No body forces are assumed to act during the process. For the discretization we use 30808 elements with 128 elements containing sides on the contact boundary. Our numerical simulation results allow various mechanical interpretation related to the contact boundary conditions, including the hardening and softening process of the foundation, as illustrated in Figure 2.

4. Conclusion

The paper deals with a newly formulated frictionless contact model for viscoplastic materials with normal compliance and unilateral constraint. The novelty of the model is the fact that the stiffness coefficient is allowed to depend on the history of the penetration. The problem is highly nonlinear and, mathematically, it is represented by a system of integral-differential equations. The main contribution of the paper is the mathematical analysis and the introduction of an efficient numerical method to solve the model.

References


The second order shape - topological differentiability of elastic energy in domains with cracks

Jan Sokolowski\textsuperscript{1}, Antoni Zochowski\textsuperscript{2}

\textsuperscript{1} Institut Elie Cartan, Laboratoire de Mathématiques Université Henri Poincaré Nancy I, B.P. 239, 54506 Vandoeuvre lès Nancy Cedex, France
and Systems Research Institute of the Polish Academy of Sciences
e-mail: Jan.Sokolowski@univ-lorraine.fr
\textsuperscript{2} Systems Research Institute of the Polish Academy of Sciences
Newelska Str6, 00-447 Warsaw, Poland
e-mail: Antoni.Zochowski@ibspan.waw.pl

Abstract

In the paper the theoretical foundations for the shape-topological sensitivity analysis of elastic energy functional in bodies with nonlinear cracks and inclusions are presented. The obtained results can be used to determine the location and the shape of inclusions which influence in optimal way the stress intensity factors at crack-tips in two spatial dimensions. The non-penetration contact conditions are prescribed on crack sides. The shape-topological sensitivity analysis of the associated variational inequalities is performed for the elastic energy functional. In variational problems the singular perturbations are replaced by regular perturbations of bilinear forms which are supported on the manifold \( \Gamma_\epsilon = \{ |x - \bar{x}| = \epsilon \} \) with \( \epsilon > 0 \). The expressions for topological derivatives thus obtained are useful in passive control of crack propagation.

Keywords: frictionless contact, elastic bodies with cracks, variational inequality, shape functional, shape sensitivity, topological sensitivity, domain decomposition, Steklov-Poincaré operator

1. Introduction

Topological derivatives of shape functionals were introduced for linear elliptic boundary value problems. The obtained formulas are given by expressions depending on point-wise values of solutions as well as of their gradients. Therefore, the expressions for topological derivatives are not well defined on the energy spaces associated with boundary value problems. In the paper we propose equivalent expressions for the topological derivatives for variational inequalities derived by a domain decomposition technique. Such expressions are given by line integrals in two spatial dimensions, or by surface integrals in three spatial dimensions. In addition, the new expressions are well defined in energy space. In order to derive the topological derivatives by an application of the domain decomposition technique the artificial interface \( \Sigma \subset \Omega \) is introduced and \( \Omega := \Omega_1 \cup \Sigma \cup \Omega_2 \) is decomposed into two subdomains.

For the boundary value problem under consideration such a decomposition is useful indeed. In some applied problems we are interested in the influence of singular perturbations in subdomain \( \Omega_1 \) on the behaviour of solutions in subdomain \( \Omega_2 \). The functional under consideration is the elastic energy \( E(\Omega) \) of a whole domain \( \Omega \). The mixed shape-topological or topological-shape second order derivatives of energy are evaluated. The shape sensitivity analysis is performed in \( \Omega_2 \), the asymptotic analysis is performed in \( \Omega_1 \). In the framework of shape-topological sensitivity analysis the method is applied to the energy functional in order to determine another shape functional \( J(\Omega) := dE(\Omega; V) \), where \( V \) is the specific vector field in derivation of \( V \rightarrow dE(\Omega; V) \). Then the asymptotic expansion of \( \epsilon \rightarrow J(\Omega_\epsilon) \) is obtained, where \( \Omega_\epsilon \) contains small inclusion or void \( \omega_\epsilon \). In the framework of topological-shape sensitivity analysis, first the asymptotic expansion of \( \epsilon \rightarrow E(\Omega_\epsilon) \) is performed, and the first order term of such an expansion is called the topological derivative. It turns out that the topological derivative of energy functional is unbounded in the energy space of the elasticity boundary value problems considered. Therefore, we study the equivalent representation of topological derivatives well defined in the energy space. These representations can be used to modify the state equations replacing the singular domain perturbations by the regular perturbations of bilinear forms in variational setting.

The asymptotic analysis of the energy functional performed in one subdomain, e.g., \( \Omega_1 \), can be used in the second subdomain \( \Omega_2 \) by means of the asymptotic expansion of the Steklov-Poincaré operator on the interface. The method is justified by the fact that the first order expansion of the energy functional in the subdomain leads to the first order asymptotic expansion of the Dirichlet-to-Neumann mapping on the interface between subdomains. Thus the first order expansion of the Steklov-Poincaré operator on the interface for the second subdomain is obtained. This way the first order expansion of the energy functional in the truncated domain \( \Omega_2 \) is derived. Precision of the obtained expansion is sufficient [7] to replace the original energy functional by its first order expansion, provided the obtained expression is well-defined in energy space. Furthermore, the first order approximation of the energy functional in \( \Omega \) is stated.

2. Domain decomposition

The proposed domain decomposition method is important for variational inequalities since the asymptotic analysis for variational inequalities is more involved compared to linear elliptic boundary value problems. The variational inequality under consideration results from the minimization problem of quadratic functional

\[
v \mapsto I(v) = \frac{1}{2} a(v, v) - L(v)
\]

over a convex, closed subset \( K \subset H \) of the Hilbert space \( H \) called the energy space. The function space \( H := H(\Omega) \) is a

\[251\]
Sobolev space which contains the functions defined over a domain \( \Omega \subset \mathbb{R}^d, \ d = 2,3 \). A singular geometrical perturbation \( \omega \) (void) centred at \( \bar{x} \in \Omega \), the size of perturbation is governed by a small parameter \( \epsilon \rightarrow 0 \). The quadratic functional defined on \( H := H(\Omega) \) becomes
\[
v \to I_\epsilon(v) = \frac{1}{2} a(v,v) - L_\epsilon(v)
\] (2)

with the minimizers \( u_\epsilon \in K := K(\Omega_\epsilon) \).

The expansion of associated energy functional
\[
e \to \mathcal{E}(\Omega_\epsilon) := L_\epsilon(u_\epsilon) - \frac{1}{2} a(u_\epsilon, u_\epsilon) - L_\epsilon(u_\epsilon)
\] (3)
is considered at \( \epsilon = 0 \). We are looking for its asymptotic expansion
\[
\mathcal{E}(\Omega_\epsilon) = \mathcal{E}(\Omega) + \epsilon \mathcal{E}(\bar{x}) + o(\epsilon^3),
\] (4)
where \( \bar{x} \to \mathcal{T}(\bar{x}) \) is the topological derivative. We show that there are regular perturbations of bilinear form defined on the energy space \( H(\Omega) \),
\[
\epsilon \to b(\epsilon, v, v)
\]
such that the perturbed quadratic functional defined on the unperturbed function space \( H(\Omega) \)
\[
v \to \Gamma^\epsilon(v) = \frac{1}{2} a(v,v) + \epsilon^2 b(v,v) - L(v)
\] (5)
furnishes the first order expansion (4). In our applications to contact problems in linear elasticity it turns out that the bilinear form \( \epsilon \to b(\epsilon, v, v) \) is supported on \( \Gamma^R := \{ x - \bar{x} = R \} \subset \Omega \) with \( R > \epsilon > 0 \).

Contact problems in elasticity are modelled using variational inequalities. It is known that the solutions to variational inequalities are Lipschitz continuous with respect to the shape. In general, the state governed by a variational inequality is not Fréchet differentiable with respect to the shape. For a class of variational inequalities described by the unilateral constraints in Sobolev spaces of Dirichlet type the metric projection onto the constraints turns out to be Hadamard differentiable. This property is used in order to obtain the first order directional differentiability of the associated shape functionals.

In order to show the second order shape differentiability for variational inequalities, we restrict ourselves to energy-type shape functionals. The energy functional is the so-called marginal function and it is Fréchet differentiable with respect to the shape. The first order shape derivative of the energy functional in the direction of a specific velocity vector field is considered as the shape functional for topological optimization. Thus, its topological derivative is evaluated.

The possible applications of shape-topological derivatives include the control of singularities of solutions to variational inequalities by insertion of elastic inclusions far from the singularities.

We consider the domain decomposition method for purposes of the shape-topological differentiability of energy shape functionals. First, the domain \( \Omega \) is split into two subdomains \( \Omega_1, \Omega_2 \) and the interface \( \Sigma \). The differentiability with respect to small parameter of the Dirichlet-to-Neumann map which lives on the boundary \( \Sigma \subset \partial \Omega_1 \) is established. This map is called the Steklov-Poincaré operator for subdomain \( \Omega_2 \).

Once the derivative of the energy functional is given, we proceed with the subsequent topological optimization problem. For topological optimization another decomposition \( \Omega := \Omega_2 \cup \Gamma_R \cup \Omega_\epsilon \) is introduced. The small inclusion \( \omega_\epsilon \) centred at the origin \( \bar{x} := \emptyset \) is located in subdomain \( \Omega_\epsilon \subset \Omega \) with the interface \( \Gamma_R \subset \partial \Omega_R \).

The singularities of the elastic displacement field at the crack front are characterized by the shape derivatives of the elastic energy with respect to the crack shape in the framework of Griffith’s theory. For example, in two spatial dimensions, the first order shape derivative of elastic energy functional evaluated in the direction of a velocity field supported in an open neighbourhood of one of crack tips is called the Griffith’s functional, which has a form similar to energy. The Griffith’s criterion was extended to the nonlinear crack models in the paper [3].

The passive control of the propagation of crack front in an elastic body is performed by the optimum shape design technique. To this end, the Griffith’s functional is minimized with respect to the shape and the location of small inclusions in the body. The inclusions are located far from the crack. In order to minimize the Griffith’s functional over an admissible family of inclusions, the second order directional, mixed shape-topological derivatives of the elastic energy functional are evaluated to determine the locations of inclusions.

Thus the domain decomposition technique is applied to both the shape and topological [7] sensitivity analysis of variational inequalities.

The knowledge of directional derivatives of the Griffith’s functional makes it possible a sensitivity analysis for the control of crack propagation in brittle materials in the context of where the Griffith’s criterion applies.

![Figure 1: Domain \( \Omega \) with crack](image)

**References**


Analysis of the impact of selected anthropometric parameters on the kinematics of the patellofemoral joint

Ewa Stachowiak1, Zbigniew Pilecki2, Alicja Balin3

1,3 Biomechatronic Department, Faculty of Biomechanical Engineering, Silesian University of Technology Rooseveltta 40, 41-800 Zabrze, Poland
e-mail: ewa.stachowiak@polsl.pl 1

2 Chorzów Pediatrics and Oncology Center
Władysława Truchana 7, 41-500 Chorzów, Poland

Abstract

Complications after stabilisation of the patella are an important orthopaedic issue. In the paper the impact of patella height on the kinematics of the patellofemoral joint was studied. Firstly the anthropometric parameters were determined on MRI scans for two groups of youth: patients with patella instability (case group) and healthy people (control group). The received parameters were analysed and used for further studies. A 3D dynamic model of the knee was created, and patella displacement was studied.

Keywords: patellar instability, patella tracking, joints kinematic

1. Introduction

The patella plays a significant role in the kinematics of the lower limb. Its main function is the improvement of efficiency of the extensor forces in the whole motion range of the knee, mainly as a result of centralization and increase of the quadriiceps force. In addition, it helps stabilize the joint and protects its front side.

The anatomical structures stabilizing the patella include the medial patellofemoral ligament, lateral patellofemoral ligament, patellar tendon and quadriceps muscle. Other factors influencing the stability of the patella are the limb alignment and joint geometry.

The currently available literature contains numerous anthropometric indices determined in order to assess the stability of the patella [1,5]. The research of the patellofemoral joint biomechanics most often includes analyses of patella kinematics, stresses occurring on the joint surfaces, extensor forces [3].

In spite of the progress in medical science and less and less invasive surgery techniques, the medial patellofemoral ligament reconstruction surgeries serving the purpose of patella stability recovery often result in complications (up to 25%) [5,6].

The considerable post-surgery complication ratio, including mainly condyle dysplasia [7], was the inspiration for the presented study. As the proper patella motion trajectory in the trochlea plays a major role in the biomechanics of the patellofemoral joint, the main aim of the research was to assess the influence of the position of the patella on the joint kinematics.

Most of the currently available literature focuses on either biomechanical or anthropometrical analyses. In the paper the patella kinematics was analysed depending on the parameters of its position. The anthropometric parameters were determined on the basis of the imaging examination, which were implemented in a model of the patellofemoral joint.

2. Materials and Methods

In the presented study an anthropometrical analysis of patella indices and creation of 3D dynamic knee model were performed.

2.1. Anthropometrical measurement and statistical analysis

MRI studies of patients from Chorzow Pediatrics and Oncology Center (ChPaOC) were taken as input data for analysis. Case group consisted of 15 youth aged from 14 to 20 years (mean: 15.7, STD: 2) – 10 women, 5 men. Patients in this group were treated in ChPaOC in 2014 and diagnosed with patellar instability. Control group consisted of 15 youth aged from 13 to 19 years (mean:16.6, STD: 1.73) – 8 women, 7 men. These patients were treated in ChPaOC in 2014 for reasons other than patellofemoral problems.

Among numerous anthropometric indices available in the current state of art, the parameters most often used by orthopaedists for assessment of patella stability were chosen: Insall-Salvati ratio, Blackburne-Peel ratio, sculus angle lateral patella displacement and the length of the patellar articular surface. Anthropometric measurements were performed in Mimics Innovation Suite, which is certified as software for medical applications.

The results were analysed with the use of the t-Student test (independent two sample t-test), which is recommended for statistical tests with a small number of samples.

2.2. Dynamic model of the knee joint.

The second stage of the research was the creation of the 3D dynamic model of the knee joint. A geometrical model based on magnetic resonance imaging of the physiological joint was built in Mimics Innovation Suite software. The geometry included: femur, tibia, patella and articular cartilage. The surface model was discretised into finite elements and then converted to volume mesh. After this step, the geometry was exported to MADYMO Software.
In the MADYMO biomechanical significant ligaments were also included: lateral collateral ligament, medial collateral ligament, anterior cruciate ligament, posterior cruciate ligament, medial patellofemoral ligament, lateral patellofemoral ligament and patellar tendon. Their mechanical properties were implemented based on literature data [4] and these elements were modelled as non-linear tensile objects. Bones were modelled as rigid bodies, and cartilages as linear elastic deformable elements.

Femur was fixed, and the patella was loaded by the resultant quadriceps force. Values of these forces were taken from model studies performed with the use of motion analysis system Xsens and Anybody software in Biomechatronic Department on Silesian University of Technology. Values of the quadriceps force during gait cycle are presented in the graph below (Fig 1). During the performed simulations the ratio of patella height (length of patellar tendon) was modified in a range of Insall-Salvati ratio from 1.0 to 1.6 and lateral displacement of the patella was examined.

Figure 1: Resultant quadriceps force during gait cycle

3. Results

The results of the anthropometric analysis are presented in table 1 (tab. 1). Based on the results of the statistical analysis, the most statistically significant parameters were determined. The greatest values of the level of significance were observed for BP and IS (where p<0,001), which are directly related to the patella height.

<table>
<thead>
<tr>
<th>Table 1: Anthropometric parameters of study groups</th>
<th>Case group</th>
<th>Control group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean</td>
<td>STD</td>
<td>Mean</td>
</tr>
<tr>
<td>Insall-Salvati ratio (IS)</td>
<td>1.422</td>
<td>0.175</td>
</tr>
<tr>
<td>Blackburne-Peel ratio(BP)</td>
<td>0.648</td>
<td>0.246</td>
</tr>
<tr>
<td>Sculus angle(SA)</td>
<td>141.49</td>
<td>11.107</td>
</tr>
<tr>
<td>Lateral patella displacement (LPD)</td>
<td>10.99</td>
<td>6.33</td>
</tr>
<tr>
<td>Length of the patellar articular surface (SL)</td>
<td>27.91</td>
<td>12.61</td>
</tr>
</tbody>
</table>

For this reason the analysis of the impact of the patella height on the patellofemoral joint kinematics was performed in MADYMO software. With the increasing height of the patella, a decreasing joint stability was observed. Also changes in patella trajectory in comparison to the normal [2] trajectory were noted. Based on the performed dynamic analyses, it was found that with the increasing patella height, the lateral patella dislocation also increases (with the same load value for all cases), which is undesirable. The displacement occurred in a range from a few mm in case of a healthy joint to almost 2 cm in case of patella alta.

4. Conclusion

This paper shows that the correct position of patella during MPFL reconstruction plays a vital role. During surgery it is necessary to take into account not only lateral patella displacement, but also patella height. Model studies could suggested the validity of using the compensation of tibial condyle in appropriate cases as the method for restoring the proper trajectory of the patella motion.

In the further studies the assessment of the impact of other parameters connected with the joint morphology (such as sculus angle, lateral patella displacement, mechanical properties and attachment points of ligaments) will be performed.

References

Stochastic approach in the modelling of implants used in hernia repair in the context of uncertainties of abdominal wall properties

Katarzyna Szepietowska1, Izabela Lubowiecka2, Benoit Magnain3, Eric Florentin4

1,2 Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: katszepi@pg.gda.pl1, lubow@pg.gda.pl2
3,4 INSA Centre Val de Loire, Université d’Orléans, PRISME EA 4229, 88 Boulevard Lahitolle, CS 60013, F18022 Bourges cedex, Bourges, France
e-mail: benoit.magnain@insa-cvl.fr3, eric.florentin@insa-cvl.fr4

Abstract

The study addresses the issue of hernia repair, which is a common medical problem. Due to high variability of properties of abdominal wall, whose mechanics is crucial in the context of hernia repair, a stochastic approach is proposed in the modelling of implant – abdominal wall system. Displacements of tacks due to daily activities are assumed as random variables representing variation in abdominal wall properties in a population. Further optimisation of hernia repair should be performed with the use of stochastic approach due to significant influence of properties of abdominal wall on forces in joints connecting implant to human tissue.

Keywords: biomechanics, nonintrusive stochastic finite element method, implants, hernia

1. Introduction

Ventral hernia is a common medical issue requiring surgery treatment. There is 12% chance that abdomen surgery will result in hernia as a postoperative complication Ref. [1]. Nowadays the gold standard of hernia treatment is implanting surgical meshes. However relapse of hernia still happens and reduction of recurrences rate is highly anticipated Ref. [9]. The necessity to optimise hernia repair results in the need of employing mechanics.

Studies on implants used in hernia repair can be found in the literature e.g. Ref. [10]. Physical and mathematical models of implant – abdominal wall systems were created too, Ref. [13]. In the context of ventral hernia mechanics of abdominal wall is crucial Ref. [4]. Studies on properties of abdominal wall e.g. Ref. [8] show great variability of these properties. High uncertainties of material properties are common in biomechanical modelling. Sensitivity study including variations in the material properties in cervical spine was studied in Ref. [5]. Identification of muscle properties including variability of experimental population was done with the use of Monte Carlo method in Ref. [11].

High variability of properties of abdominal wall leads to conclusion that stochastic approach is relevant in the modelling of hernia repair. In the study models of implant- abdominal wall system are developed basing on Nonintrusive Stochastic Finite Element Method. Although, in this group of methods appropriate choice of sampling points is crucial, the optimal way to do so is not fully Ref. [2] determined. Different approaches can be reviewed in the field Ref. [3, 6]. In the work preliminary outcomes of employing nonintrusive stochastic approach will be presented to model implant-abdominal wall systems on the example of implant subjected to extortions caused by daily activities. Attention is paid to the forces in fasteners, because recurrences are usually caused by joint failure.

2. Model

Mathematical model of implant Ref. [7] (Fig. 1) is based on Finite Element Method (Fig. 2). It is built of membrane 8-node quad elements for an orthotropic bilinear elastic material model Ref. [13] simulating an implant. In the location of fasteners translations are restrained. Implant is subjected to displacement of fasteners (kinematic extortions) simulating displacement of fasteners during human activities.

Figure 1: Scheme of implanted surgical mesh subjected to tacks displacements

Figure 2: Scheme of implanted surgical mesh subjected to tacks displacements

*This study is partially supported by the subsidy for young scientists given by the Faculty of Civil and Environmental Engineering, Gdańsk University of Technology. Computation were performed partially in TASK Computer Science Centre, Gdańsk, Poland.
3. Results

The results are presented in the abstract. Figure 3 shows a histogram of maximum reaction forces obtained from Monte Carlo simulation ($10^5$ simulations) for variable changes in abdominal properties imposed to first three fasteners.

![Histogram of maximum reaction forces (forces in fasteners)](image)

In the sample case of hernia placement, type of implant and orientation the maximum reaction in the model calculated for the mean properties of abdominal wall is equal to 13.4 N, the mean value due to MC simulation is 14.9 N, whereas the 95% percentile is equal to 18.8 N.

4. Conclusions

The first results show the necessity of employing stochastic approach in this particular biomechanical application. Design of the number of fasteners or optimal properties of implant taking into account only deterministic simulation with average parameter can lead to dangerously underestimated value of forces in fasteners in the case of properties of abdominal wall not being mean values. Including the 95% percentile value in design allows solutions suitable for a wide range of populations, required to take into account difficulties in identification of properties of abdominal wall for every patient before repair and patient-specific design of implant and other hernia repair parameters.

References


On various modelling approaches to real-time visualisation of blood flow

Krzysztof Tesch¹, Katarzyna Kaczorowska²

¹,²Faculty of Mechanical Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: krzyte@pg.gda.pl

Abstract

The paper reviews various modelling approaches to real-time visualisation of blood flow. These include classic, macroscale approach based on the momentum conservation equation together with a proper constitutive equation. Moreover, modern micro- and mesoscale approaches, such as molecular dynamics and dissipative particle dynamics, are discussed. Advantages and disadvantages of the discussed methods are highlighted with particular attention to real-time visualisation. A fast method of blood flow visualisation is introduced.

Keywords: blood flow, CFD, micro- meso- and macroscale approach

1. Introduction

Blood is typically regarded as a non-Newtonian fluid due to the presence of red blood cells in the plasma. Blood motion may be described by three types of mathematical models according to the observed scales: microscopic description (molecular dynamics), mesoscopic description (e.g. dissipative particle dynamics) and macroscopic description (classical fluid mechanics). All of these methods have advantages and disadvantages. The microscopic approach has a high computational cost. To reduce these computational demands, the mesoscale models simulate only a reduced number of degrees of freedom. Finally, the macroscopic approach results in averaged information.

2. Macro-, micro- and mesoscale descriptions

Although blood is a suspension of blood cells in the plasma, blood flow in large vessels is regarded as a single component and single phase fluid. The non-Newtonian phenomena, such as the shear thinning, yield stress and constant viscosity values at high shear rates are modelled by means of a proper constitutive equation.

The closed system of equations for laminar, incompressible and non-Newtonian fluids consists of the continuity Eqn (1a), and the linear momentum conservation Eqn (1b). The system of four scalar equations, from the macroscopic point of view, appears as follows:

\[ \nabla \cdot \mathbf{u} = 0, \]

\[ \rho \frac{d\mathbf{u}}{dt} = \rho \mathbf{g} - \nabla p + \nabla \cdot \mathbf{\tau}. \]

In the above, \( \mathbf{u} \) denotes the velocity vector, \( \mathbf{g} \) stands for the acceleration due to gravity, \( p \) indicates pressure, \( \rho \) density and the viscous part of the stress tensor is denoted as \( \mathbf{\tau} \).

Another, the so-called rheological constitutive equation is needed to close the above system. We divide rheological constitutive equations into categories of Newtonian-, generalised Newtonian-, differential-, integral- and rate type fluids (Ref. [1]). Generalised Newtonian fluids satisfy the following rheological equation \( \mathbf{\tau} = 2\mu(\gamma)\mathbf{D} \). The second invariant of the strain rate tensor \( \mathbf{D} \) is given by \( \gamma^2 = 2\mathbf{D}^2 \). For a differential type of fluids the viscous part of the stress tensor \( \mathbf{\tau} \) is expressed explicitly as a function of other kinematic tensors and their derivatives \( \mathbf{\tau} = f(A_1, A_2, \ldots) \) whereas for the rate type fluids this equation is not explicit \( \mathbf{\tau} = f(\mathbf{\tau}, \mathbf{D}, \mathbf{D}) \). The dot represents the frame-invariant derivative.

The generalised Newtonian fluids are the simplest and easiest to implement into existing CFD codes (Ref. [6]). More advanced models such as differential- and rate type fluids are able to better approximate blood features but they cannot be directly implemented into commercial CFD codes. In addition, even the most advanced constitutive equations cannot satisfy all the attributes of blood. Furthermore, the macroscale approach based on the momentum conservation equation is not particularly suitable for the real-time visualisation of blood flow. Although, the macroscopic approach is able to simulate blood flow for large space and time scales (big arteries), it breaks down entirely in the capillaries. This is because in vessels with a diameter smaller than 1 mm non-Newtonian properties of blood are important. Explicit modelling of red blood cells is required for arteries of diameters smaller than about 100 – 200 µm.

Figure 1: The simplest representation of red blood cells as rigid spheres

Molecular dynamics method, being a fundamental microscale method, belongs to the so called discrete particle methods group. The method takes advantage of classical mechanics equations to model molecular systems in the context of N-body simulation.
The motion of individual molecules is determined by solving the Newton’s equation of motion

\[ m \frac{d^2 \mathbf{r}_n}{dt^2} = \mathbf{G}_n + \sum_{j=1,j \neq i}^{N} \mathbf{f}_{ij}. \]  

(2)

The force exerted on a molecule consists of the external force such as gravity \( \mathbf{G}_n \), and the intermolecular force \( \mathbf{f}_{ij} = -\nabla V \) usually described by means of the Lennard-Jones potential \( V \). The ensemble average makes it possible to obtain a macroscopic quantity from the corresponding microscopic variable. The disadvantage of molecular dynamics is that the total number of molecules even in a small volume is too large, meaning that only small volumes can be modelled. Methods of periodic boundary conditions are utilised to overcome this limitation.

Dissipative particle dynamics, being a mesoscale method, is able to overcome difficulties and limitations of molecular dynamics method and still preserves some molecular details that capture the physics of blood. The method may be used when the continuum description is not appropriate and the molecular dynamics approach is not manageable due to system size. The mesoscale models simulate only a reduced number of degrees of freedom. These are the so-called coarse-grained models. This simply means that the continuum is discretised into mesoscale particles being larger than atomistic scale. The mesoscale particles are regarded as clusters of atoms. The motion of particles is determined by solving the Newton’s equation of motion

\[ m \frac{d^2 \mathbf{r}_n}{dt^2} = \mathbf{G}_n + \sum_{j=1,j \neq i}^{N} \left( \mathbf{f}_{ij}^{C} + \mathbf{f}_{ij}^{D} + \mathbf{f}_{ij}^{R} \right) \]  

(3)

where the interaction forces are the sum of conservative or repulsion forces \( \mathbf{f}_{ij}^{C} \), dissipative forces \( \mathbf{f}_{ij}^{D} \) and random force \( \mathbf{f}_{ij}^{R} \).

Both, micro- and mesoscale approaches or their combination with macroscopic models are suitable for real-time blood visualisation. This is particularly possible for the so-called passive transport where the walls of the vasculature are considered to be rigid. Fig. 1 displays real-time numerical simulation results of the in-house made code. The diameter of the tube is 260 μm which corresponds to a small artery. An average diameter of the human red blood cell 8 μm is assumed. The bulk flow is regarded as a macroscopic non-Newtonian flow according to the Ostwald-de Waele model \( \mathbf{\tau} = 2k \mathbf{\dot{e}}^{\text{max}} D^{-1} \mathbf{I} \). If vessel diameters are larger than 100 μm, continuum approach of blood flow still provides a good approximation. This is true, however, provided that a non-Newtonian model is adapted. Furthermore, analytical solutions of the bulk flow in a circular pipe can be easily obtained which means that the bulk velocity boundary conditions can be imposed. Movement of red blood cells is modelled by means of a particle-based approach according to Newton’s second law. The presented approach allows for a real-time visualisation of more than \( 10^{4} \) RBCs on a modern CPU (above 30 fps). The method can be further extended to account for more realistic shapes for red blood cell. This can be achieved by means of the fixed connection of particles allowing for deformation, see Fig. 2.

Other methods of coupling the red blood cell membranes with a fluid domain resolved by the lattice-Boltzmann allowing for active transport and cells deformation are given in Ref.[5, 3, 2]. The individual vertex \( n \) of the triangulated surface of the red blood cell moves according to

\[ M \frac{d^2 \mathbf{r}_n}{dt^2} = \mathbf{f}_{n}^{FS} + \mathbf{f}_{n}^{PP} + \mathbf{f}_{n}. \]  

(4)

The external forces due to the fluid-structure interaction on the vertex are denoted as \( \mathbf{f}_{n}^{FS} \). By \( \mathbf{f}_{n}^{PP} \) one understands forces due to particle-particle interaction. Forces due to the Helmholtz free energy contribution are calculated as

\[ \mathbf{f}_{n} = -\frac{\partial E}{\partial \mathbf{x}_n}. \]  

(5)

The Helmholtz free energy \( E \) consists of a sum of the in-plane, bending, volume and area contribution. Finally, \( M \) stands for a fictitious mass at each vertex.

The multiscale approaches that couple mesoscale and macroscale models are discussed in Ref.[4, 7].

References

Influence of the stiffening effect on mesh fixation in ventral hernia repair

Agnieszka Tomaszewska1, Izabela Lubowiecka2, Czesław Szymczak3

1,2 Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mails: atomas@pg.gda.pl 1, lubow@pg.gda.pl 2
3 Faculty of Ocean Engineering and Ship Technology, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: szymce@pg.gda.pl

Abstract

The problem of experimental and numerical modelling of abdominal wall – fastener – implant system built in laparoscopic ventral hernia repair is discussed in the paper. An important issue of the implant stiffness change under load history is addressed. The goal is to check the influence of the stiffening effect on deflection of the system under impulse pressure load simulating post-operative cough of a patient as well as its influence on the junction force in the mesh fixation points. Junction force values are the key results, as the mesh fixation is designed to carry that force. The research is conducted for the selected implant and the example geometry of operated hernia. It proves that, due to the stiffening effect, the considered material may find itself in two states: initial (flexible) and preconditioned (stiff). Similar displacements of the considered model have been obtained in the numerical and experimental approaches. As it was assumed, the displacements are smaller for the preconditioned state than for the initial state of the implant, whereas the junction forces have higher values. That proves the accuracy of the physical modelling of the selected medical case as well as of the material model identification and FEM modelling.

Keywords: ventral hernia, biomechanics, FEM modelling, experiment

1. Introduction

The paper refers to laparoscopic ventral hernia repair made with the use of knit synthetic mesh. Such meshes are commonly used in medical practice. Although laparoscopic procedures have been developed for four decades obstacles still occur, e.g. hernia recurrences and chronic post-operative pain happen [3]. Mechanical calculations concerning behaviour of a system built of abdominal wall – fastener – mesh are conducted in order to improve medical outcomes. One of the recent observations in the field refers to the stiffening effect of knit synthetic meshes. The phenomenon is observed in cyclic loading and unloading tests [1,5]. For the time being mechanical calculations for the operated hernia systems are conducted based on material models identified in simple tensile tests, the stiffening effect is not considered. In such an approach the junction force calculated for the mesh fixation points may be underestimated comparing real condition e.g. the fact, that the junction force value increases with the implant stiffness increase [4]. The issue is particularly important, because the mesh fixation is selected basing on value of the junction force. Our team is the first one that considers the stiffening effect influence on the mesh junction force. The research is made for the DynaMesh®-IPOM (FEG Textiltechnik mbH, Germany).

2. Material and methods

2.1. Implant material modelling

In numerical simulations the implant is modelled by means of dense net material model. Such approach is positively verified in the paper [2]. Constitutive equations for two axes of the model are to be identified based on uni-axial tests of the material samples. This is the way of the research. Uni-axial tensile tests and cyclic loading and unloading tests are made using Zwick Roell 2020 testing machine. The device is equipped with a video extensometer and is calibrated for the force range 0÷20 kN. Rectangular samples with dimensions of 120×30 mm have been prepared. They have been cut in two perpendicular directions of the implant; one of the directions is the most compliant one. Stress-strain experimental curves have been specified based on measured forces and extensions. Engineering measures for stress and strain are used. It has been observed in the cyclical tests that the material stiffness increases within successive load loops, that is the stiffening effect mentioned before. Thus, two states are distinguished for the material: initial (before loading) and preconditioned (material reloaded). Earlier research proved that the mesh may achieve both states many times [5]. For the two states and the two directions of the mesh bilinear constitutive functions are determined. For the initial state stress-strain functions obtained from simple tensile tests are a basis for the constitutive functions identification. The basis for the constitutive function identification in the preconditioned state are relations describing material stiffness change within successive loading in the cyclic tests. Details of the identification are discussed in the paper [5]. The following pairs of elasticity moduli values describe each constitutive model. For the initial state of the implant for its most compliant direction the following values are obtained: 367 and 3353 N/m with a value change for strain 0.45, whereas for the perpendicular directions the values are 1309 and 5920 N/m and the value change is for strain 0.2. For the preconditioned state of the implant in its most compliant direction the values are 1060 and 5788 N/m with a change when strain equals 0.45, whereas in the perpendicular directions the values are 2087 and 6000 N/m with the strain value changed at 0.15.
2.2. Local modelling of operated hernia

A study on dynamic behaviour of operated hernia has been performed using a specially prepared experimental stand shown in Fig. 1.

The surgical mesh implanted to the porcine abdominal wall has been installed in the pressure chamber. The impulse load of air pressure has been applied and the displacements of implant and joints have been registered by laser sensors. The experiment has been performed in two steps assuming that the implant material stiffness is changing in the preconditioned state. In the initial stage the pressure grew up until the value of 7.75 kPa within 6.5 s, while in the second stage the pressure grew up until 6 kPa within 3.75 s. The experiment scheduled in this way delivered the information necessary to verify the simulation results with the identified initial and preconditioned constitutive functions of the material and thus to confirm the stiffening of the implant.

A numerical model of the considered implant-tissue system has been defined using finite element method as described e.g. in [2]. It is a local model where the implant is defined by membrane finite elements and the tissue part is implemented in the form of appropriate boundary conditions. The elastic foundations represent the abdominal wall supporting implant and the surrounding fascia is defined by elastic springs. Elastic foundations stiffness assumed as 2.7e6 Pa refers to the abdominal wall stiffness and the spring stiffness set as 1.5 kN/m refers to the fascia stiffness. The model has been previously calibrated and verified on the basis of the experimental data presented in [2]. The 19 tissue-implant joints have been defined in the model and hernia orifice diameter of 7 cm has been assumed as realised in the experiment. The material model described in Section 2.1 has been applied to the simulations.

Numerical simulations have been performed representing the two stages of the experiments as described before with the initial and preconditioned stiffness of the surgical mesh. The model reaction forces represent here the forces in tissue-implant joints.

### Table 1: Numerical vs. experimental displacement of implant [mm]

<table>
<thead>
<tr>
<th></th>
<th>Initial state (7.75 kPa)</th>
<th>Preconditioned state (6.00 kPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Max deflection Orifice edge</td>
<td>Max deflection Orifice edge</td>
</tr>
<tr>
<td>Experiment</td>
<td>31.5 14.3</td>
<td>24.5 9.6</td>
</tr>
<tr>
<td>Simulation</td>
<td>32.5 12.2</td>
<td>26.3 8.44</td>
</tr>
</tbody>
</table>

3. Results and discussion

Simulations have been performed applying different stiffness functions of the considered implant, in its two states: initial and preconditioned. Displacements of the implant obtained in simulations and experiments are similar. However, in the preconditioned state the displacements are smaller. This results from the pressure lower than in the first simulation and also from the stiffening of the mesh. Since the junction forces at upon the hernia repair persistence (see e.g., [2]), the maximum reactions in the model supports have been also calculated within the simulations. In the initial state, $R_{\text{max}}^0 = 0.41$ N and in the preconditioned state $R_{\text{max}} = 0.51$ N. Even if the pressure is lower in the second simulation and experiment (preconditioned state), the junction force is higher. This clearly confirms the importance of the mesh stiffening considerations and shows that it affects the junction forces.

4. Conclusions

The study deals with the mechanical behaviour of the implanted surgical mesh used in hernia repair under impulse pressure referring to the patients cough. Two considered stages of the implant loading in experiment and in simulation represent the initial and preconditioned state of the implant material. The initial refers to the first mesh loading, the preconditioned to the following pressure impulse. The stiffening effect observed in experiments has been confirmed by numerical simulations. It is an important phenomenon since it results in the increase of the maximum junction forces in the tissue-implant connection. Thus, it can easily lead to the hernia repair failure even if the initial stiffness of the surgical mesh does not indicate the junction forces exceeding the tearing force value specified for the implant.

References


Numerical realization of boundary integral equation method for solving two-dimensional non-stationary problems of thermoelasticity

Andrei Verameichik\textsuperscript{1}, Vitaly Garbachevsky\textsuperscript{2}, Vitaly Khvisevitch\textsuperscript{3}, Valery Rakhuba\textsuperscript{4}

\textsuperscript{1,2,3,4} Brest State Technical University
st. Moskovskaja 267, 224017 Brest, Belarus
e-mail: vai_mrtm@bstu.by

Abstract

A FORTRAN coded computer programme was developed on the basis of the integral equations boundary method. The programme is intended for calculating the elements of construction that are in the conditions of plane stress or plane deformation. This sphere can be of mono- or multi-tier type. The developed programme may be used to solve both inner and outer boundary tasks. For visual presentation of the ‘FORTRAN’ calculation results their processing was made with the help of Tecplot 360 graphic interface. A number of problems of investigating stressed-deformed state of construction elements were solved with the help of the devised programme.

Keywords: thermoelasticity, boundary element method, numerical solution

1. Introduction

The topicality of the question of simultaneous influence of heat and mechanical loads on the elements of constructions is explained by complexity of geometric forms of these structural elements and the character of the influences mentioned. With a view to reduce components cost price while preserving their durable and rigid characteristics for the investigated stressed and deformed state it is offered to use the method of boundary integral equations \cite{1,2,4,6,7}. The use of numerical method for solving such kind of problems is explained by the impossibility of solving analytically the differential equations (DE), describing the given process, under complex boundary conditions. In practical terms, considering such problems it is sufficient to limit the investigations considering a plane area \cite{2}, that is why the paper considers a two-dimensional non-stationary problem of thermoelasticity \cite{3}.

2. Solution Algorithm

On the basis of numerical algorithm of solving boundary problem integral equations \cite{7,8} a ‘FORTRAN’ computer programme was developed for investigating stressed-deformed state of isotope bodies during mechanical or temperature loading. The programme is intended for calculating constructive elements in the conditions of flat strained state or flat deformation. The area occupied by a body can be of mono- or multi-tier type. On the basis of this programme it is possible to solve both inner and outer boundary problems. In this case, on the boundary of a body there must be set of a balanced outer load vectors only for those parts which cannot be substituted for a circumference or straight line. The programme envisages the diagnostics of the input and processing initial data. When preparing the data it should be taken into account that a boundary or inner point cannot be closer than 0.5 of the

their lengths, curvature radii and outer perpendicular vector ordinates. At this stage the body of outer load is formed.

At the second stage is solved the system of linear algebraic equations, the result of which are the values of potential densities in the points of the area boundary. An algebraic system is solved by the method of Gauss exceptions. Matrices are not fully preserved in memory. Their processing is done by lines. When forming matrix lines, depending on the distance between a parameter point and integration piece there is determined the number of Gauss’ quadrature point knots.

The third stage obtains stresses and transferences in the boundary area in the corresponding inner area points \cite{7}.

When building the calculated area it is necessary to strive to obtain a boundary with piece-continuous, limited curvature. If, for any reason, it is difficult to bring the problem to smooth boundary, the programme envisages a possibility of realizing the area with angles (prominent and in-going knots). In this case the problem is solved for functioning equation in which the contour \(L\) in each angle point widens by two pieces \(AB\) and \(BC\) up to the contour \(L_1\) (fig. 1).

In a standard procedure the programme realizes the tasks for the areas limited by pieces of straight lines and circumferences. The area under consideration can be optional, set graphically or analytically. In connection, changes are introduced into the sub-programme of initial data input and contain sub-programmes or set of operators for calculating the ordinates of the centres of the pieces of the boundary fragmentation, curvature radii, lengths, outer load vectors only for those parts which cannot be substituted for a circumference or straight line. The programme envisages the diagnostics of the input and processing initial data. When preparing the data it should be taken into account that a boundary or inner point cannot be closer than 0.5 of the

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{figure1.png}
\caption{Widening of the area with angle}
\end{figure}
length of the piece of the boundary fragmentation \( \Delta_k \) to the parametrical point.

Except using natural symmetry planes, an artificial plane is often introduced, alongside with it, not losing the peculiarities of the problem being solved. The introduction of artificial planes in most cases can considerably facilitate the actions on the preparation of initial data.

3. Results

As test tasks for perfecting the authenticity of the developed algorithm, the problem of thermoelasticity for a chisel was solved. The given detail is used to punch holes. The material for the chisel is steel Sverker 21. The area is loaded at the punch end with evenly distributed contour load 100\( \cdot \)10^6 N/m, and has only outer contour. The chisel has a symmetry plane. The stresses at the fixed end can be presented as an even contour load. Taking into account the symmetry peculiarities, the calculated scheme for the realization of this problem assumes the following view as given in fig 2.

As a result of the realization of the problem the stress state in the corresponding points was obtained (fig. 3).

A comparison was made of the boundary element analysis results with the results of numerical solution by means of finite element method carried out using computing complex ANSYS, and also with the results of theoretical calculation [3]. As a result of the carried-out numerical experiments to evaluate the stressed-deformed state one can make a conclusion that the stress areas obtained with the help of ANSYS packet and the boundary integral equations method do not differ qualitatively (fig. 4).

In the places of the changes of a body geometrical form (potential density function breaks) the stress concentration is observed. The quantitative stress estimate showed that the results of solving the problems with the help of these methods differ within admissible limits when making engineering calculations. As a result of comparing the possibilities of these methods it should be pointed out that the use of the ‘FORTRAN’ programme significantly simplifies the preparation of initial information.

References

Application of artificial neural networks in man’s gait recognition

Tomasz Walczak¹, Jakub K. Grabski², Magdalena Grajewska¹, Martyna Michalowska⁴

¹,²,³,⁴ Institute of Applied Mechanics, Poznan University of Technology
Jana Pawła 24, 60-965 Poznań, Poland

Abstract

In the paper a method of human gait recognition based on artificial neural networks is presented. The method is based on observation, that each individual has different, unique way of basic movement activity. Hence, parameters may be defined to represent typical characteristics of the gait of each person. In a classical approach the video analysis is used to obtain the set of sequential pictures of a moving person. Then, a similar 2D-stick model is applied, to represent the gait signature and to describe typical kinematic characteristics. In this paper, all considered gait parameters are obtained from data of dynamometric platforms used to measure the acting forces during the gait. These gait characteristics are unique for each person, and some of geometrical qualities of curve which represents vertical component of force, are taken into the account. In order to implement the recognition process the back-propagation neural network algorithm is used. In experiments, higher gait recognition performances was achieved. It will be proved, that it is possible to recognize person, based on gait characteristic parameters taken from dynamometric platforms.

Keywords: biomechanics, gait parameters, artificial neural networks, man’s gait recognition

1. Introduction

The gait may be defined as a coordinated, cyclic combination of movements, that result in human locomotion [3]. Human gait has common patterns of movements, coordinated to have a specific temporal pattern for the gait to occur. These movements are repeated as man cycles between steps with alternating feet. It means, gait can be described as a periodic motion.

A period of the gait cycle exists between the successive heel strikes, and the gait motion in space and time, satisfies spatial and temporal symmetry. In a similar way, other kind of motions may be described, e.g. running, jogging or climbing stairs. Moreover, human motion analysis includes many challenging issues, because of a highly flexible structure of the human body. Human gait is known to be one of the most universal and complex from all human activities. Thus, the study of human gait has increased extensive interests in various fields such as clinical analysis, biomechanics, robotics, and biometrics [2, 3, 8, 10].

Human gait is usually described by kinetic or kinematic parameters, which are obtained from a simple mechanical model, where all considered human parts are considered pendulums, of a simple harmonic motion. Those parameters describe some gait patterns. Each individual appears to have different and unique characteristics of gait. It means, that it is possible to identify person on the base of the gait parameters [4,5,8].

That kind of recognition of individuals could be also used in clinical practice to identify improper or pathological gaits.

In this paper, the method for recognizing humans by their gait using artificial neural network (ANN) is presented. To describe and recognize characteristics of gait for each considered individual, untypical approach was examined. In classical way, the pattern of gait description is based on video analysis, where obtained sequences of pictures from camera are processed to 2D stick human body model. Then some necessary kinematic parameters, like trajectories and velocities of characteristic points or characteristic angles, are determinate. That kind of analysis is complex requiring use of many time-consuming algorithms.

2. Neural network

The artificial neural networks are widely used for many problems of recognition, prediction or control processes [1,2,6,7,9]. Algorithms based on ANN are effective and could process a lot of data in a relatively short time. For that reason they are commonly used in recognition problems, where usually the pattern to classification are found. The neural network methods facilitate gait analysis because of their highly flexible, inductive, non-linear modeling ability, unlike any other approaches. The non-linear property of multi-layered neural networks is useful for the analysis of complicated gait parameters, which traditionally have been difficult to model in an analytical way.

To recognize the gait, back-propagation algorithm for training multi-layered neural network, based on selective retraining and a dynamic adaptation of learning rate and momentum, is implemented. The pattern recognition system contains an input subsystem, that accepts sample pattern vectors and a decision-maker subsystem, that decides the classes to which an input pattern vector belongs [4]. The scheme of multi-layered neural network structure, used to gait recognition, is presented on Figure 1.
measurements allows to define some parameters, that well describe gait for each individual. Typical shapes of curves, that represent those forces are presented on Figure 2.

![Figure 1: The scheme of used neural network](image1)

The network architecture presented in Figure 1 consists of input layer, one or two inner layers and output layer. The function $f$ is defined for every neuron in the system as neuron activation function with S-shaped characteristic, vector $a$ contains parameters, that describe gait, and vector $s$ is an answer vector, that contains response of the network in recognition process, where $s_i$ component is a network response responsible for $i$-th recognizing individual. That architecture should allow not only recognition of the individuals set, but also show similarity of the gait of the considered persons.

The Lavenberg-Marquardt method will be applied to train network, and no more than 50% of data collected for each individual will be used during the learning process of the network.

### 3. Gait identification parameters

The proposed method of human gait recognition is based on data obtained from measurement of force, where feet act on dynamometric platforms. High sensitivity and accuracy of those measurements allows to define some parameters, that well describe gait for each individual. Typical shapes of curves, that represent those forces are presented on Figure 2.

![Figure 2: Vertical component of force for both feet, measured during gait](image2)

Note, that for most cases the curves representing vertical component of forces have always two maxima and one local minimum. However some curve geometric parameters could be completely different for each considered individual. Some important parameters are:

- Static moments of curve.
- Differences between the value of maximum and minimum.
- Distance between both maxima.
- Area between curve and time axis.
- Difference between force measured in the case of left and right foot.

Obviously, before determination of all mentioned parameters, all data should be normalized. It allows to determine some patterns for each person and should lead to effective recognition process with use of neural network. It will be proved, that it is possible to recognize person, based on gait characteristic parameters taken from dynamometric platforms.

### References


Interactive application using virtual reality technology for supporting of diagnostic process of upper limbs

Piotr Wodarski1, Robert Michnik2, Jacek Jurkojć3, Andrzej Bieniek4, Marek Gzik5

1, 2, 3, 5 Biomechatronic Department, Faculty of Biomedical Engineering, Silesian University of Technology, Akademicka 2A, 44-100 Gliwice, Poland
E-mail: piotr.wodarski@polsl.pl, robert.michnik@polsl.pl, jacek.jurkojc@polsl.pl, marek.gzik@polsl.pl

4 Faculty of Biomedical Engineering, Silesian University of Technology, Akademicka 2A, 44-100 Gliwice, Poland
E-mail: andrzej.a.bieniek@gmail.com

Abstract

Virtual Reality technology recently finds its new application becoming very popular among diagnostics and rehabilitation systems. There are a lot of applications dedicated to lower limbs, but still only a few for upper limbs. A proper performance of these systems requires a motion capture system allowing for determination of kinematic and dynamic data of body motion.

In the research an attempt to create and develop a system supporting the upper limb diagnostic processes was made. The research was conducted by scientists from Silesian University of Technology (Biomechatronics Department from Faculty of Biomedical Engineering). A physiotherapist from Jan Paweł II Pediatrics Center in Sosnowiec and a group of 25 student participated in the study. Three-dimensional diagnostic applications in virtual reality cave system were prepared, with a scenario to perform selected exercises according to the selected pattern. Using the inertial motion analysis suite MVN Biomech two tasks, carried out by each of 25 subjects from a group of students, were recorded three times. The results of the research include normalized courses of joint angles in upper limbs.

The calculations of loads in human musculo-skeletal system were performed using AnyBody system. In this way, it is possible to determine reaction forces and moments in joints as well as forces generated by muscles.

Keywords: virtual reality, diagnostics of upper limb, interactive application, rehabilitation

1. Introduction

Virtual Reality technology systems have become very popular as supporting tools used in diagnostic systems, where Motek Medical system is one of the most widespread dedicated for the lower limb rehabilitation [1]. Up till now, there were a lot of attempts to create systems supporting rehabilitation of upper limbs similar to Motek. An example of a system for upper limb motion was described in Sandeep research [2], using Virtual Reality Technology to create virtual object. These object might were movable in a limited space.

Determination of kinematic and dynamic data of an upper limb motion can be done using motion capture systems [3]. Optical or accelerometer motion capture systems are ready for a parallel registration and determination of angle quantities in time for all joints [4]. The recorded values are input data for mathematical models to analyse loads in human skeletal-muscular system. The developed methodology may be implemented to therapeutic and diagnostic processes.

2. Material and method

The research was conducted by scientists from Silesian University of Technology (Biomechatronics Department from Faculty of Biomedical Engineering). A physiotherapist from Jan Paweł II Pediatrics Center in Sosnowiec (age 25 year) and a group of 25 student (mean age 23,3 year SD 1,34) also participated in the study. There were no defects of the upper limbs in subjects. Two upper limb motion sequences were selected according to the motions performed during diagnosis and rehabilitation process of children in real clinical conditions. Figures 1 and 2 show sequences of the selected motions.

Using an inertial motion analysis suite MVN Biomech three normal patterns of movement sequences performed by a physiotherapist were recorded. On the basis of recorded motions three-dimensional diagnostic applications (were prepared in fact being funny games), dedicated for the cave system. While playing a game, they require to perform the recorded rehabilitation exercises. The elaborated scenarios motivate to perform selected exercises according to rehabilitation patterns.

Figure 1: The sequence of motion no. 1

Figure 2: The sequence of motion no. 2

The two selected applications, with their scenarios, were tested by students (Fig. 3). Each student had to play a game three times, the movements were recorded by means of the MVN Biomech system.
3. The result

The measurements were done to determinate joint angles in upper limbs during selected exercises. Exemplary courses of angles in glenohumeral joint are presented in Fig. 6. The obtained results were the input data for the mathematical model of the upper limb prepared in AnyBody system. The prepared model was presented in Fig. 5. This model was used to calculate forces generated by muscles and joint reactions (exemplary results in Fig. 4, where the values of reactions were standardized). The results obtained for the physiotherapist were compared with these obtained for students.

4. Discussion and Conclusion

The application implemented in Cave system allows to increase motivation to perform selected exercises. Similar to Sandeep [2] things are created in virtual world where different scenarios would include various exercise programs. The developed methodology involving the transformation of recorded data comes from the motion analysis system. The created transformation allows to determine loads in skeletal-muscle system in AnyBody program. Thus, it is possible to determine reaction forces and moments in the joints and muscle strength values for each implemented single muscle. The recorded kinematic data and calculated dynamic data are implemented in a database. The database of developed models with correct motion patterns serve as a standard reference for people with mobility problems in the upper limbs. The study is the first step to create an automatic system using virtual reality technology for supporting diagnosis of upper limb processes.

References


Thermal properties of biomaterials on the example of the liver

Ryszard Wojnar¹, Barbara Gambin²*

¹,² Department of Ultrasound, Institute of Fundamental Technological Research of the Polish Academy of Sciences
Pawiińskiego 5B, 02-106 Warszawa, Poland,
e-mail: rwojnar@ippt.pan.pl¹, bgambin@ippt.pan.pl²

Abstract

Lionel Smith Beale, FRS, (1828–1906), a physician and microscopist in an evocative comparison wrote that the liver resembles a magnificent tree with its trunk and branches, with myriad of leaves, synthesizing and detoxifying. The liver in a human is about the size of football, equipped in a circulatory system and is made of about one million primary lobules which are almost identical, like the leaves of the tree. Therefore, the liver from mathematical point of view can be considered as a micro-periodic medium, and the mathematical methods of homogenisation developed for micro-periodic media can be applied to determine some overall properties of the tissue. Pennes equation of heat propagation in a biological tissue is a quasi-nonlinear partial differential equation with coefficients depending on temperature T. It consists of three terms, one of them describes Fourier heat diffusion, with the diffusion coefficient λ depending on T. This term is a subject of the contribution.

Keywords: Pennes equation, micro-periodic structure, effective conductivity

1. Introduction

After the discovery of ultrasound generators it was realized that absorption of high intensity ultrasound waves acts negatively on biological tissues. This observation led to research in tissue heating and healing effects. However, there is still little information on the effect of heating on absorption by tissues, which affects the size and shape of the thermal lesions. The absorption coefficient exhibited by a soft tissue varies widely from tissue to tissue and is a function of temperature, cf. [7].

The liver is composed of four lobes of unequal size and shape, with a rich micro-structure. A normal human liver weighs about 1.5 kg. The liver is a vital organ with a wide range of functions including protein synthesis and storage, transformation of carbohydrates, synthesis of cholesterol, bile salts and phospholipids, detoxification, and production of biochemicals necessary for digestion, [2, 3].

A hepatic lobule (Lat. lobuli hepatis) is a small division of the liver defined at the histological scale. It is about 1 million lobules in the human liver, each lobule containing at least 1000 sinusoids 0.5-1.0 mm in length, and 700 nm in breadth. There are over 1 billion sinusoids, with blood sluggishly flowing in parallel through each one.

A hepatocyte is a cell of the main parenchymal tissue of the liver. Hepatocytes make up 70-85% of the liver mass. The typical hepatocyte is similar to a cube with sides of 20-30 μm.

A liver sinusoid is a type of sinusoidal blood vessel (with fenestrated, discontinuous endothelium) that serves as a location for the oxygen-rich blood from the hepatic artery and the nutrient-rich blood from the portal vein. Sinusoidal capillaries are a special type of open-pore capillary also known as a discontinuous capillary, that have larger openings (30-40 μm in diameter) in the endothelium, [10].

2. Acoustic wave

George Döring Ludwig (1922 - 1973), pioneer in medical ultrasound, estimated the velocity of sound transmission in animal soft tissues between 1490 and 1610 m/s, with a mean value of 1540 m/s. He also determined that the optimal scanning frequency of the ultrasound transducer was between 1 and 2.5 MHz, and found that the speed of ultrasound and acoustic impedance values of high water-content tissues do not differ greatly from those of water, [12].

Acoustic wave propagating through a fluid in the direction x with the speed u; amplitude A and angular frequency ω is described by

\[ \xi = A \sin \omega \left( t - \frac{x}{u} \right) \]  

Here \( \xi \) denotes the displacement of the particle in the time t at point x. The velocity of vibrations \( v \equiv \partial \xi / \partial t \) has the amplitude \( v_0 = A \omega \). The amplitude of strain \( e_0 \equiv \partial \xi / \partial x \) is \( e_0 = A \omega / u = v_0 / u \). From Newton’s equation we have

\[ \rho \frac{\partial^2 \xi}{\partial t^2} = - \frac{\partial p}{\partial x} \]  

where \( \rho \) is the density of the fluid and \( p = p(x, t) \) is the acoustic wave pressure. Hence

\[ \frac{\partial p}{\partial x} = A \rho \omega^2 \sin \omega \left( t - \frac{x}{u} \right) \]  

and \( p = p_0 + A \rho \omega u \cos \omega \left( t - \frac{x}{u} \right) \). The integration constant \( p_0 \) denotes the pressure in the fluid in absence of wave. The pressure variation is \( p = p_0 + A \rho \omega u \cos \omega \left( t - \frac{x}{u} \right) \), with the amplitude

\[ p_0 \equiv p(t = 0) = p_0 \]  

The stream of the energy \( \mathcal{J} = wu \), where \( w \) denotes the mean value of the wave energy

\[ w = \frac{1}{2} \rho v_0^2 \]  

After time averaging (\( \langle \ldots \rangle \) \( = (1/t) \int_0^t (\ldots) d\tau \)) we get, so called, the acoustic pressure

\[ p^* \equiv \langle p_0 \rangle = w = \frac{\mathcal{J}}{w} = \frac{1}{2} \left( \frac{\mathcal{J}}{\rho u^2} \right)^2 \]  

*This work was partially supported by the National Science Centre (grant no. 2011/03/B/ST7/00347).
In a soft biologica tissue an acoustical wave with a speed 1500 m/s and frequency \( \nu = \omega/(2\pi) = 1 \text{ Mhz} \) has the length 1.5 mm. Let the acoustic pressure \( p^* \) be 1 N/m\(^2\) = 1 J/m\(^3\). Then \( v_0 = (2/3) \cdot 10^{-16} \text{ m/s} \), and the amplitude of vibrations \( A = (2/3) \cdot 10^{-3} \text{ nm} \), what is a sub-atomic length.

3. Attenuation of sound

Due to a sound propagating, there is always thermal loss of energy caused by viscosity. For inhomogeneous media, besides viscosity, acoustic scattering is another reason for the removal of acoustic energy, \([5]\).

Absorption of ultrasound in biological tissues strongly depends on the molecular composition of the tissue. The absorption coefficient increases in function of a protein content, with collagen of a particularly high specific absorption. Collagen accounts for 10% in the liver, and the absorption coefficient is of 0.2 dB/cm. In water and body liquids there is little absorption 0.003 dB/cm, \([1]\).

4. Thermal properties and Pennes equation

The heat in a living body is transferred by three different mechanisms: conduction, convection (natural or forced) and radiation. Average thermal conductivity of the liver in (W/(m \( \cdot \) K)) is 0.52 with a standard deviation 0.03. The same numbers are found for the blood, \([6]\). The Pennes equation reads, \([9]\),

\[
\rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x_k} \left( \lambda \frac{\partial T}{\partial x_k} \right) + w_b c_b (T - T_a) + q
\]

(7)

Here, \( \rho \) and \( c \) are the mass density \([\text{kg/m}^3]\) and specific heat \([\text{J/kg} \cdot \text{K}]\), respectively, \( w_b \) is the blood perfusion rate \([\text{m}^3/\text{blood} / \text{s} \cdot \text{kg tissue}] \). \( T_a \) is the temperature of the arterial supply blood.

The heat generation term \( q \) encompasses the thermal effects of metabolism and, if necessary, other volumetric heat loads, as microwave irradiation or the heat generated by ultrasound waves.

5. Nonlinearity of thermal properties

There is a considerable variation in thermal properties from tissue to tissue, from species to species, and even within tissues from the same donor. Thermal properties of water taken from \([11]\) were fit to a linear equation over the range 0\(^\circ\)C to 45\(^\circ\)C, it is \( \lambda = 0.5652 + 0.001575 T \) where \( \lambda \) is in (W/(m \( \cdot \) K)). Thermal conductivity of a tissue is lower than that of water, while the temperature dependence approximates that of water, it is \( \lambda = 0.4882 + 0.001265 T \). Thermal diffusivity of a tissue matches the thermal diffusivity of water well for both the magnitude and the temperature coefficient.

6. One-dimensional time-independent problem

Let the section of \( x \geq 0 \) of \( x \) axis consist of two sub-sections (segments) \( x_1, x_0, x_0 + \ell \) and \( x_0 + a, x_0 + \ell \). Let the heat conductivity be \( \lambda_a(1 + \alpha T) \) in \( x_0, x_0 + a \) and \( \lambda_0(1 + \beta T) \) in \( x_0 + a, x_0 + \ell \).

For small \( \alpha \) and \( \beta \) the temperature \( T_a = T(x_0 + a) \) is given by

\[
T_a = \frac{1}{b\lambda_0 + a\lambda_0} \left( b\lambda_a(T_0 + \frac{1}{2}aT_0^2) + a\lambda_0(T_0 + \frac{1}{2}\beta T_0^2) \right)
\]

(8)

In the linear case, for \( \alpha = 0 \) and \( \beta = 0 \) we have

\[
T_a = \frac{1}{b\lambda_0 + a\lambda_0} (b\lambda_a T_0 + a\lambda_0 T_0)
\]

(9)

For \( \alpha > 0 \) and \( \beta > 0 \) the expression (8) is always greater than \( 9 \).

7. Effective thermal conductivity of liver

Effective medium approximations describe a medium (composite material) based on properties and relative fractions of its components. These approximations include a Clausius-Mossotti’s formula (CMF) for the effective conductivity of the medium consisting of the matrix substance of conductivity \( \lambda_M \), in which small spherical inclusions of conductivity \( \lambda_i \), are disseminated the ratio of the volume of all small spheres to that of whole being \( f \).

\[
\lambda_{eff} = \frac{\lambda_i + 2 \lambda_M + 2(\lambda_i - \lambda_M) f \lambda_i}{\lambda_i + 2 \lambda_M - (\lambda_i - \lambda_M) f \lambda_M}
\]

(10)

Vladimir Mityushev defined the effective thermal conductivity when the conductivity coefficient is a function of the temperature \( T \), and found a generalization of CMF for a family of strongly non-linear and weakly inhomogeneous composites, \([8]\). Unfortunately, in the liver tissue the inhomogeneity of conductivities (collagen vs fluid) is strong.

A. Galka, J. J. Telega and S. Tokarzewski noticed that soft tissues are usually anisotropic, and using asymptotic methods derived the formula for the effective heat conductivity in this general case, \([4]\).

In one-dimensional case their formula reads

\[
\lambda_{eff} = A(\xi) + B(\xi) + \frac{C(\xi)}{T - D(\xi)}
\]

(11)

where \( \xi = \alpha / T \), cf. Sect. 6, and \( A, B, C, D \) are given function of \( \xi \). Their effective conductivity \( \lambda_{eff} \) is no more linear in the temperature, however. This formula is applied to describe the \( \lambda_{eff} \) of the liver.

References


\([6]\) http://www.itis.etbi.ch/itis-for-health/tissue- properties/database/thermal-conductivity/

\([7]\) Kurglenko, E., Gambin, B., Cieliski, L., Soft tissue-mimicking materials with various number of scatterers and their acoustical characteristics, Hydroacoustics, 16, pp. 121-128, 2013.


Modelling polycrystalline structure of collagen fibrils dense packing by the most uniform concentric pattern

Ryszard Wojnar
Institute of Fundamental Technological Research Polish Academy of Sciences, IPPT PAN, Pawiińskiego 5B; 02-106 Warszawa, Poland
e-mail: rwojnar@ippt.pan.pl

Abstract

Tropocollagen molecules (TC ~300 nm long) spontaneously self-assemble, with regularly staggered ends, into larger arrays known as fibrils. In most abundant fibrillar collagens of type 1, TC are staggered from each other by about 67 nm, a distance that is referred to as D-period and changes depending upon the hydration state of the aggregate. Each D-period contains approximately 4.4 TC. Therefore, there is a part containing 5 molecules in cross-section (called “overlap”) and a part with 4 TC. TC subunits are organized in such a way that in perpendicular plane of cross-section they form polycrystalline quasi-hexagonal packing (T.J. Wess 13 papers 1995-2007 and my: "Bone and cartilage–its structure and physical properties" in: "Biomechanics of hard tissues" Wiley 2010 and "Piezoelectric phenomena in biological tissues" in: "Piezoelectric Nanomaterials for Biomedical Applications" Springer 2012) with vacancies, edge dislocations, grain boundaries (GB). Graphene advances may help for the best representation of this packing.

Keywords: collagen, fibrils, packing, polycrystalline, dislocations, optimal structure

1. Introduction

John Werner Cahn in 1965 first described 5/7 edge dislocation in 5-, Hexa-, 7-gonal close-packing (5H7). Also GB and triple junctions (TJ) in 5H7 during his lecture "Euler Law and penta-hepta defects" at MIT (1970) biological conference "Shaping of tissue by deviation from hexagonal close-packing of cells" were discussed. These ideas were inspired by Bragg-Nye model of recrystallization in dense packing of small equal bubbles. This model was simulated on computer in [4]. In 1966 Francis Crick (Nobel, DNA, 1962) asked: "Superlattice of collagen is a neglected problem and it is time somebody took it up again", [2].

Figure 1: Representation of the "self-limiting" 5 : 7 subunit model proposed for the TC macromolecule. The model is based on two subunits whose lengths \( l_1 \) and \( l_2 \) are in the ratio 7:5. Each \( \alpha_1 \) strand comprises five identical subunits of length \( l_1 \); the \( \alpha_2 \) strand, seven identical subunits of length \( l_2 \). The molecular length \( L \) is the "beat period" of the two subunit lengths, \( L = 5 l_1 = 7 l_2 \). After J. A. Petruska and A. J. Hodge, 1964, [6].

Figure 2: Structure of the collagen fibril. The arrow at left represents the tropocollagen molecule (T). After [6].

In most abundant fibrillar collagen of type 1, T are shifted along each other by ~ 67 nm – referred to as D-period, joining ~ 4.4 T. Therefore, there is a part containing 5 T in cross-section (called “overlap”) and a part with 4 T called “gap”. Such 5 units cannot be in mutual contacts (like in 5-colouring) and must obey some additional local near-neighbours rules used by [1,3] for different 5H7 patterns, see Figs. 1 and 2.

2. Quasi-hexagonal packing of tropocollagen proteins

Rope-like (rod-like cylinders of length ~ 300nm) tropocollagen proteins triple-helices (T) are tightly packed together in collagen fibril (CF). The T lateral packing in perpendicular plane of CF cross-section forms 2D polycrystalline quasi-hexagonal packing [1,3]. It could be represented by 5H7 of Voronoi polygons (V) around T centers.
3. Proposed TC packing by the most uniform concentric pattern

In [9], the TC packing was proposed, performed by the most uniform concentric 5H7 pattern presented in Figs. 3 and 4. Proper understanding of it comes from the new research on very rich structure of similar 2D polycrystals.

![Figure 3: Proposed T packing by the most uniform concentric pattern: a fragment of fibril cross-section. The boundaries of hexagonal grains are completely similar to 2D crystallization.](image)

![Figure 4: Proposed fiber cross-section obtained as the most uniform concentric pattern. Every cross-section (small circle) denotes the T macromolecule, and 5 colours correspond to 5 different positions of T–molecules in fibril, depicted in Fig.2.](image)

4. Graphene analogy

The TJ of grain boundaries (GB) in graphene are recently investigated as main centers for cracks, elucidated by Boris Yakobson [8] as TJ additional heptagonal disclination, cf. also [5]. It could be similar for cracks in collagen degenerating during diseases, especially such as found recently in children’s osteogenesis imperfecta.

5. Mechanical meaning

The optimal structure of the collagen fiber should assure its mechanical strength and stability. The longitudinal staggering (Fig. 2) allows for the homogeneity in axial direction of the fiber. On the other hand, in transversal cross-section (Fig. 4), for the planar packing Euler’s law guarantees the global compensation (equal numbers of pentagons and heptagons) and enforces the local compensation (pentagon and heptagon should be nearby). These mutual compensations of positive (5) and negative (7) topological charges allow to reduce the stress concentrations inside of the fiber. The close packing of T molecules adjust in such a manner that the fiber behaves like an optimal continuous medium.

A proposed distribution of tropocollagens in the fiber cross-section is shown in Fig. 4. Mathematical constraints of 5–colouring strongly influence mechanical properties of quasi-crystalline close packing in the collagen fiber. Molecules arranged on a hexagonal lattice, with molecular D segments (corresponding to axial stagger positions) from 1 to 5 are depicted with different colours. The analogy with phyllotaxial structure, together with appearing of parastichies is evident, cf. [1,7], but the essential difference with phyllotaxis is also visible: the distribution of tropocollagens is uniform, what can assure the optimal mechanical strength of the fiber.

References


Strength analysis of sportsmen knee joint

Maciej Wykupil\(^1\), Antoni John\(^*\)

\(^{1,2}\) Mechanical Engineering Faculty, Silesian University of Technology
Konarskiego 18A, 44-100 Gliwice, Poland
e-mail: maciej.wykupil@polsl.pl\(^1\), antoni.john@polsl.pl\(^2\)

Abstract

In the paper a complex problem is presented. The main goal of the study is to analyze the strength of selected parts of the sportsman skeleton using the FEM analysis. This work focuses on analyzing the knee joint – the data necessary to carry out the simulation of effort state was obtained from the study of movement analysis for elements of the game of handball and thankfully to CT. As the results, the stress, strain and displacement distribution in numeral model of knee joint (in femur, tibia and cartilages) were calculated. Obtained results shown possibility of injury during performing game elements. On the ground of this results it is possible to modify workout in such way to reduce load acting on the sportsman body.

Keywords: knee joint, FEM analysis, movement analysis, CT data

1. Introduction

The present work combines scientific disciplines such as biomechanics, numerical methods and medicine. Here, the strength analysis of selected parts of the sportsman skeleton was performed using FEM. The main part concerns analyzing of the knee joint – the data necessary to carry out the simulation of strength was obtained from the study of movement analysis for elements of the game of handball and CT examination. The first step it was the movement analysis preparation. BTS System was applied. The performed studies allowed to determine the position of the sportsman and the value of reaction forces, which were necessary to calculate the reaction forces acting on a selected joint. In the next step numerical simulation was performed. The numerical model of a knee joint was based on the CT data. The 3D model of the femur and tibia was prepared. While having determined loads acting on the selected structure, boundary conditions were determined - it proceeded to carry out the strength analysis. Finally, the distribution of the maximum displacements, strains and stresses were calculated. The results are intended to help in the search for solutions aimed at reducing the loads acting on the selected part of the body.

2. Movement analysis

The first step of the practical part of the study was to conduct movement analysis. The main goal of this analysis was to measure ground reaction and record body (lower limb) position at the moment of maximum load. Examined five people divided into two groups: first group consist of two handball player (one male and one female), second group consist of people, who play sport in a recreation mode: one runner, one man practices karate and one juggler (all of them are men). There are considered three elements of handball game: attack (take-off), attack (landing), block. To ensure proper conduct of examination, examination protocol was prepared.

In the study, BTS Smart – motion capture system – was used. Used motion capture system tracks anatomically characteristic points’ trajectory. These data are synchronized with measurement of the ground reaction (Ref. [1,2]).

Each person participating in the study answered the questions about his sports activity and history of the injuries. Two people regularly take part in the training of handball (AZS Gliwice section). During the examination, a female practitioner handball had been struggling with ankle injury, but the injury did not have significant impact on her mobility (Ref. [3]). Three people from the entire group had gone through various injuries of lower limb.

Examination was carried out according to the Davis protocol, which is normally used in the analysis of gait. Handball players performed three game elements: attack (take-off), attack (landing), block. For others people had chosen similar elements from volleyball.

In the next step the reactions in the knee joint were calculated (Ref.[2]). It was necessary to know body position, weight and height examined person, acceleration of segments of the lower limb. Weights of individual body segments and position of their mass centre were calculated on the basis of the available literature data.

3. Numerical model and simulations

In order to create 3D model of femur and tibia the tomography data were used. A 24 years old male was subjected to CT examination (Ref. [4]). This male has suffered from pain due to knee injuries, which are results of intensive amateur sports (long-distance running, cycling, volleyball). During the study following application was:

- *MIMICS* (Materialise),
- *MSC.Patran/Nastran*,
- *HyperMesh* (*Altair-HyperWorks*),
- *CAD program allows to edit *.stl files*.

3D model of bones was created using MIMICS system (by Materialise) Ref. [5,6]. To create outline of bones, the program uses thresholding of radiation absorption. The workflow for creating the model is as follows:

1. Define the radiation absorption coefficient for a given anatomical structure.
2. Segmentation of the images of the test structure.
3. Choosing the right piece of the structure and image filtering.
4. Creating a points cloud corresponding to the contour of the desired structure, and its subsequent conversion to a surface.

\*The paper is supported by Project 10/040/BK_15/0006, 502-1100, 10-040
5. Improve the quality of the model.
7. Export the model to the CAE program’s preprocessor.
8. Creating a volumetric model.
9. Re-export to MIMIC program to purpose of assigning of material properties.

The surface mesh was created by Magic program, which is combined with MIMICS. However, obtained mesh is poor quality and it is difficult to use it in the calculation. The program creates triangle elements only (with 3 nodes). In the study Hypermesh was used to improve mesh quality.

Based of material properties follow on the basis of relation between radiological density and physics density was derived, then the relation between physics density and Young’s modulus were assumed from literature, Ref. [7,8,9]:

\r
\r
To ensure contact between both bones, it was necessary to model elements replace a cartilage, because cartilage is not formed a proper contact between radius and physics density, and next interdependencies between physics density and Young’s modulus were assumed from literature, Ref. [7,8,9]:

- constrained nodes at end section of tibia (all direction),
- contact between bones and cartilage (type Glue-compliance displacements for different mesh density),
- Total Load at nodes at end section of femur model.

Using Hypermesh, the femur model was rotated to position identified during movement analysis. For this purpose local coordinate system was created where one axis corresponds to the femur axis of rotation. It is difficult to find the right position of this axis because this axis are moving during movement of bone.

Simulations were performed for 4 positions: anatomical and three position corresponding to position of the lower limb during the examination. Due to fact, that maximum loads had occurred in the performance of attack (take-off), these loads had been chosen to analyzes. In the Tables 1 and 2 present selected results.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1083</td>
<td>-344</td>
<td>0.94</td>
<td>8.60x10^{-3} 33</td>
</tr>
<tr>
<td>2</td>
<td>1635</td>
<td>293</td>
<td>0.29</td>
<td>8.07x10^{-3} 18</td>
</tr>
<tr>
<td>3</td>
<td>2361</td>
<td>-371</td>
<td>0.45</td>
<td>1.18x10^{-2} 28</td>
</tr>
</tbody>
</table>

4. Conclusions

The preliminary results are presented here. On the basis of the obtained results finally conclusions can be written:

- The results shown possibility of injury during performing game elements, because calculated maximum stress value (220 MPa, due to individual voids in bone tissue) reached dangerous condition (stress limit).
- Stress concentration in the point of contact (in cartilage) also reached dangerous value. This results are confirmed by fact, that examination person suffers from damage to the meniscus.
- Displacement distribution results from applied boundary condition and stiffness of the model.
- The highest reduced stress values obtained for position no. 1 and no. 2. This knowledge allows to modify workout in such way to reduce load acting on the sportsman body.

The presented problem requires a larger population to movement analysis and advanced strength analysis for model with muscles, tendons and ligaments.

References

Vibrations of multi-span frames under moving load by using modified Finite Element Method

Filip Zakęś1, Paweł Śniady2

1,2 Faculty of Environmental Engineering and Geodesy, Wroclaw University of Environmental and Life Sciences
Plac Grunwaldzki 24a, 50-365 Wroclaw, Poland
e-mail: filip.zakes@up.wroc.pl

Abstract

Dynamic behaviour is studied of multi-span frames subjected to various types of moving load. Dynamic displacements of analysed structures are determined by combining modified finite element method, using known analytical solutions for single-span beam elements. The main assumption of the presented method is to divide a frame to finite elements (pinned-fixed or fixed-fixed Euler-Bernoulli beam elements) and to use known shape functions approximating its deflection line. A number of elements corresponds to the number of frame bars. A numerical example of a two-bay frame loaded with a moving point force is presented in the paper.

Keywords: frames, vibrations, moving load,

1. Introduction

In structural mechanics problems of vibrations due to moving load were analysed by many authors for many years because of their significance and practical appliance in design of various types of structures subjected to this type of load, such as bridges, roads, railways etc.. Many different structural models (single- and multi-span beams, frames, trusses, strings, plates, shells) as well as different sorts of moving load (both non-inertial and inertial) have been considered [1-7].

The paper is aimed at the analysis of dynamic behaviour of Euler-Bernoulli multi-span frames subjected to various types of moving load. The proposed method can be applied for non-inertial types of moving load such as moving point force, distributed load, concentrated moment or stream of moving forces as well as for inertial loads (moving concentrated or distributed mass, moving oscillator with one or many degrees of dynamic freedom).

2. Method description

Let us consider a multi-span Euler-Bernoulli frame shown in Fig.1. The frame consists of horizontal bars are loaded with a vertical force \( p(x - vt) \) moving with constant speed \( v \) and unloaded columns (vertical or slanting). Main geometric assumptions describing analyzed frame are listed below.

- Frame elements subjected to a moving load make a continuous horizontal line.
- Flexural stiffness \( E I \) and mass per unit length \( \mu \) are constant for each frame element, but can be different for every single bar.
- Nodes 1, 2, ..., \( k \) in which horizontal bars and columns are connected are stiff.
- Frame can be arbitrarily supported.

Such a frame can be divided into a number of finite elements which are pinned-fixed or fixed-fixed single-span beams. Shape functions \( N_{ij}(x) \) for each element resulting from the boundary conditions are used to build the equation of motion describing dynamic behaviour of the analysed structure.

\[
\mathbf{M} \ddot{\mathcal{R}}(t) + \mathbf{K} \mathcal{R}(t) = \mathbf{f}(t) \tag{1}
\]

where matrix of inertia \( \mathbf{M} \), matrix of rigidity \( \mathbf{K} \) and load vector \( \mathbf{f}(t) \) are built according to know procedures, using shape functions. Eqn (1) can be analytically solved, using eigentransformation.

After solving the equation of motion, flexural vibrations of the frame can be presented in the form:

\[
v(x,t) = \varphi_{ij}(t) \cdot N_{ij}(x) + \varphi_{ji}(t) \cdot N_{ji}(x) + w_0(x,t) \tag{2}
\]

where:

- \( N_{ij}(x) \) and \( N_{ji}(x) \) are shape functions describing deflection line of the \( ij \) element resulting from the unit rotation of each end of the element,
- \( \varphi_{ij}(t) \) and \( \varphi_{ji}(t) \) are the rotation angles at the both ends of the element determined from the Eqn (1),
- \( w_0(x,t) \) describes the vibrations of the single span element with the corresponding boundary conditions.

2.1. Equation of motion

If we take the rotation angles of nodes 1, 2, ..., \( k \) as unknowns (general coordinates), the equation of motion for the undamped vibrations of the frame shown on Fig.1 can be presented in the form:

\[
\mathbf{M} \ddot{\mathcal{R}}(t) + \mathbf{K} \mathcal{R}(t) = \mathbf{f}(t)
\]

Figure 1: Multi-span frame subjected to a moving load.

273
2.2. Vibrations of the single-span beam elements

Vibration of the uniform single-span Euler-Bernoulli beam element of a finite length \( L_{ij} \) subjected to a point force of constant magnitude \( P \) moving with the constant velocity \( v \) can be described by the equation:

\[
w_{ij}(x,t) = P \sum_{n=1}^{\infty} \frac{1}{Y_n \omega_n} W_n(x) \sin \omega_n(t - \tau) W_n(v \tau) d\tau
\]

where \( W_n(x) \) are the eigenfunctions of the beam satisfying appropriate boundary conditions, \( \omega_n \) are the eigenfrequencies and the expression \( Y_n^2 \) is equal to:

\[
Y_n^2 = \int_0^L [W_n(x)]^2 dx.
\]

Eqn (3) is true for \( 0 \leq t \leq L_{ij}/v \). When force \( P \) is leaving the beam \( t \geq L_{ij}/v \) Eqn (3) takes the form:

\[
w_{ij}(x,t) = P \sum_{n=1}^{\infty} \frac{1}{Y_n \omega_n} W_n(x) \int_0^{L_{ij}/v} \sin \omega_n(t - \tau) W_n(v \tau) d\tau
\]

3. Numerical example and results

The example presents a uniform frame with two horizontal spans of equal length and a vertical column (see Fig. 2). Flexural rigidity of all elements equal \( EI = 8.774 \times 10^6 \text{Nm}^2 \) and the same mass per unit length \( \mu = 52.4 \text{kg/m} \). The length of each element of the frame is equal to \( L_{1A} = L_{1B} = L_{1C} = 10 \text{m} \). The frame is loaded by a concentrated force of constant magnitude \( P = 10,000 \text{N} \), moving along the horizontal spans of the frame from left to right and with a constant velocity \( v = 90 \text{m/s} \).

In further analysis the frame is divided to three fixed-fixed beam elements (2 horizontal spans and the column), the rotation angle of the node “1” is considered unknown in the equation of motion, solved similarly to the single degree of dynamic freedom systems.

Diagrams on Fig. 3 and 4 display the dynamic deflection of the mid-span sections “a” and “b” compared to the influence line of static deflection at the corresponding sections.

![Figure 2: Two-bay frame subjected to a moving point force](image)

**References**


Mechanical and Thermo-Chemical Interactions in the Reactors with Moving and Fluidized Bed

organized by T. Chmielniak, D. Kardaś, S. Polesek-Karczewski and S. Stelmach
Inverse, non-destructive technique of retrieving conductivity of orthotropic materials

Wojciech P. Adamczyk¹, Ryszard A. Bialecki², Tadeusz Kruczek³*

¹,²,³ Institute of Thermal Technology, Silesian University of Technology
Konarskiego 22C, 44-100 Gliwice, Poland
e-mail: wojciech.adamczyk@polsl.pl ¹, ryszard.bialecki@polsl.pl ², tadeusz.kruczek@polsl.pl ³

Abstract

A novel non-destructive measurement technique for retrieving thermal conductivities of orthotropic bodies is presented. Developed technique combines numerical simulations with inverse method. Evaluated results compared against thermal conductivities measured using flash Parker method. A good agreement of both techniques is found one.

Keywords: inverse method, thermal conductivity, CFD, orthotropic body

1. Introduction

Thermal conductivity (TC) assessment is essential in the context of simulations of heat transfer problems, whenever heat conduction is involved. Assessing the heat losses and gains, checking admissible temperature levels of the material of construction, evaluating thermal stresses and many other engineering computations, are but just a few examples, where the knowledge of TC is indispensable. Moreover, the quality of some materials is measured by their heat conductivity. Thermal insulation, carbon blocks used for blast furnaces are examples of such situations. Practically, the only way, TC values can be obtained are experiments. The current state of the art for the TC measurements, the available techniques, their advantages and disadvantages are described elsewhere eg. [1,2,3,4,5], and the overview will not be repeated here. Desired features of TC measurement techniques are their non-destructive character and rapidness. Within the great number of available TC measurement techniques, only very few can be applied in-situ ie., without extracting probes of the material under investigation. Most of available techniques can be directly applied only to materials of isotropic structure, which limits applicability of these methods.

The paper presents a non-destructive, rapid technique applicable to anisotropic (orthotropic) materials. Retrieving material properties belongs to a wide class of inverse techniques whose three components direct technique of solving heat transfer problem, experimental method of measuring temperature fields and optimization procedure of adjusting the simulated temperature field to the measured one. The developed method uses computational fluid dynamics to solve the direct problem, records the spatial and temporal variation of the temperature generated by a laser pulse and solves optimization problem by Levenberg-Marquardt procedure. The technique is an extension of earlier works of the research team, where the direct problem has been simulated using a simple, analytical model based on Green’s function of semi-infinite adiabatic medium heated by Dirac impulse [6]. Though the heat losses from the boundaries have not been included in the model, the proposed technique gives the possibility of accounting for the heat losses due to natural convection and radiation from the heated surface, which is more realistic than the assumption of ideal insulation thereof. Besides, the technique allows to retrieve all components of the orthotropic TC tensor, while the analytic technique yielded only the in-plane two components.

2. Experiment

The test rig built at the Institute of Thermal Technology is shown in Fig. 1. As the source of energy high power laser IPG Photonics was used, whose maximum power amounts to 200 W, with the smallest duration of the laser pulse less than 0.05 s. The temperature field was recorded by IR FLIR camera (resolution 320x240, 60 fps), placed perpendicularly to the observation plane. Orthogonal location of the laser beam with respect to the observation plane, reduced the error associated with non-circular shape of the heat source and temperature measuring error of the IR camera. As opposed to the standard Parker method, both the heat source (laser) and the temperature measuring device (IR camera) are located on the same side of the observation plane. The measuring process, as well as data acquisition were controlled using in-house PC application written in National Instrument LabVIEW code.

![Figure 1: Experimental rig](image)

3. Calculation strategy for orthotropic materials

The technique is dedicated to orthotropic samples of hexahedral shape with principal axis of the TC aligned with the principal direction of the sample. For such configuration three principal components of the TC tensor are retrieved by fitting the simulated to measured temperatures. The idea of measurement procedure is highlighted in Fig. 2. The laser heating and IR temperature recording is subsequently applied to x = z and y = z planes. After recording in the first plane the sample is rotated around by 90 degree and the heating-recording

* The research was supported by National Science Centre within Opus scheme, under contract UMO_2013/11/B/ST8/00268.
process is repeated. For each plane, the same measurement procedure is used: recording initial temperature field, laser emission, recording changes of the temperature field. In the inverse procedure, the data from both recordings are fitted to the numerical model with components of the TC tensor treated as decision variables.

Figure 2: Calculation strategy

The mathematical formulation of the objective function for orthotropic material assumes form

\[
\min \sum_{i,j} \left( \left( \Theta_{x_iy_j} \right) (x_i, y_j, T_0) - \Theta_{x_iy_j} (x_i, y_j, T_n) \right) \]

where \(N\) defines the number of degrees of freedom, \((x_i, y_j)\) stand for spatial coordinates of the \(i\)-th pixel centre, and the ratios of experimental and numerical temperature differences are defined as

\[
\Theta_{x_iy_j}(x, y, T_0) = \frac{T(x_i, y_j, 0) - T_i(x_i, y_j, T_0)}{T(x, y, 0) - T_i(x, y, T_0)}
\]

where \(T_0\) is the initial time, \(T_n\) represents the subsequent times where measurements were taken. The above relations assume similar form for \((z-y)\) plane. The heat conduction problem in the sample was solved using commercial finite volume solver with the laser energy modelled as a heat source employing user defined function. The sensitivity matrix used by the Levenberg-Marquardt method was defined as the set of first derivatives of the simulated excess temperature with respect to the Levenberg-Marquardt method was defined as the set of first derivatives of the simulated excess temperature with respect to the material specific heat and density were equal 900 J/kgK and 1091 kg/m3, respectively. The characteristic values of the laser flash in the developed inverse technique were: emission time 0.19 s and power 67.4 W. The propagation of temperature field after laser emission is illustrated in Fig. 3. The calculated thermal conductivities for selected material are collected in Table 1.

Table 1: Numerical results

<table>
<thead>
<tr>
<th>(\lambda_x), W/mK</th>
<th>Inverse</th>
<th>Measurement</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.39</td>
<td>5.3</td>
<td></td>
</tr>
<tr>
<td>4.06</td>
<td>4.6</td>
<td></td>
</tr>
<tr>
<td>5.58</td>
<td>5.4</td>
<td></td>
</tr>
</tbody>
</table>

5. Conclusions

The presented measurement technique for retrieving thermal conductivities of three main components of TC tensor of orthotropic material shows satisfactory accuracy when compared data obtained using standard, destructive measurement method. The technique may be readily extended to handle more realistic boundary conditions (i.e., accounting for heat losses due to radiation and natural convection, yielding better higher accuracy. Generalization to arbitrary shapes and anisotropy is a topic of current research. The main disadvantage when using the proposed approach the long evaluation time, which excludes the technique form integrating it with online measurements performed at the production line. The work on applying reduced order methods to circumvent these difficulties in underway.

References

Numerical modelling of CO$_2$ enriched gasification of coal in a pressurized circulating fluidized bed reactor

Joanna Bigda$^{1,6}$, Adam Klimanek$^2$, Tomasz Chmielniak$^3$, Wojciech Adamczyk$^4$, Andrzej Szcł$^5$, Sławomir Stelmach$^6$

1,3,6 Institute for Chemical Processing of Coal
Zamkowa 1, 41-803 Zabrze, Poland

2,4,5 Institute of Thermal Technology, Silesian University of Technology
Konarskiego 22, 44-100 Gliwice, Poland

Abstract

In the paper preliminary simulation results of CO$_2$ enriched gasification of coal in a pressurized circulating fluidized bed are presented. Two numerical models of a pilot scale gasifier were developed using commercial codes ANSYS FLUENT and CPFD Barracuda have been used. The numerical models encompass the barrel like section and the riser of the gasifier. The separator and downcomer have not been included. Coal pyrolysis, homogeneous reactions and heterogeneous reactions are considered. Flow patterns, as well as the local particle velocities, particle solid fractions and gas product composition are obtained. The results provide a detail insight into gasifier behaviour including fluidization, thermal and chemical characteristics. The differences between the two numerical models are presented. Simulated outlet gas composition using two numerical models is compared with experimental data and a quite good resemblance is observed.

Keywords: coal gasification, pressurized circulating fluidized bed, CFD modelling

1. Introduction

Various approaches are applied to model particle laden flows and in particular the hydrodynamics of circulating fluidized beds. A frequently applied methodology can be viewed as meso-scale, in which the spacial scales cover the characteristic sizes ranging from particle size to the sizes of the modeled device [1]. The approaches in this group include the Eulerian-Eulerian model, in which the gaseous and solid phase are treated as interpenetrating continua, and Eulerian-Lagrangian models in which the gaseous phase is treated as continuous and the solid phase as discrete. It should be stressed that in the Eulerian-Lagrangian models the particles or their groups (clouds) are tracked in the Lagrangian frame of reference. Effective tracking of particle groups allows modeling of systems containing large numbers of particles typical for pilot and industrial scale [2,3]. The Lagrangian models, applicable to dense particle laden flows, implemented in commercial codes ANSYS Fluent and CPFD Barracuda belong to this group, although they differ in several aspects. Both approaches have already been used in modelling coal gasification in fluidized bed reactors [4,5]. A particularly challenging in this respect is coupling of the complex unsteady multiphase flow and chemical (heterogeneous and homogeneous) reactions. Although many models have been developed to account for the turbulence-chemistry interaction in gaseous mixtures, no such model is available today to account for the interaction of turbulence and the heterogeneous reactions on reacting particles. In the paper a simple chemical/diffusion model is used to account for the rate of reaction at particle surface. Although such an approach is a crude approximation of the real behaviour, it is a computationally inexpensive alternative to more complex models. The gaseous phase homogeneous reactions are modelled using the Eddy Dissipation Concept approach available in Fluent. The CPFD methodology is based on the MP-PIC method, which uses a stochastic particle method for the particle phase and an Eulerian method for the fluid phase to solve equations for dense particle flow. An enthalpy equation describes energy transport for fluid and provides for transfer of sensible and chemical energy between phases and within the fluid mixture. Homogenous and heterogeneous chemistry are described by reduced-chemistry and the reaction rates are implicitly solved numerically on the Eulerian grid. Inter-phase momentum and energy transfer are also implicitly calculated, giving a robust numerical solution from the dilute flow to close-pack limits [5].

2. The experimental facility

The experimental facility modelled in this study is a pilot scale pressurized circulating fluidized bed reactor built in Institute for Chemical Processing of Coal in Poland. The reactor encompasses a barrel like bottom part, in which internal recirculation of the solid phase develops and a riser section is connected to a separator and recirculation section. Geometry of the reactor used to create the numerical model is presented in Figure 1. The coal is introduced at the side of the barrel part and the gasifying agent at the very bottom of the reactor through a mesh inlet. The unreacted char can be circulated to the bottom part of the reactor however in the investigated case it was not.

The gasifying agent in this study is a mixture of CO$_2$, O$_2$ and N$_2$. Thermodynamic calculations presented in [6] revealed that addition of CO$_2$ to the gasifying agent can lead to significant decrease of fuel consumption and increase of CO production capacity when compared to conventional (air or oxygen blown) gasifiers.

3. Numerical models

The numerical model applied in this study is based on the previously developed approach presented in [4]. Some...
improvements introduced to this model include: application of the eddy dissipation concept with Jones-Lindstedt four step mechanism for the gaseous phase reactions. The model developed in Barracuda is based on the approaches presented in [5].

3.1. Model input data

The main input data used as boundary conditions in the model are summarized in Table 1.

Table 1: Main model input data

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coal mass flow rate, kg/h</td>
<td>32.46</td>
</tr>
<tr>
<td>Gasifying agent mass flow rate, kg/h</td>
<td>72.04</td>
</tr>
<tr>
<td>Gasifying agent composition (mass fractions):</td>
<td></td>
</tr>
<tr>
<td>CO₂</td>
<td>0.68</td>
</tr>
<tr>
<td>O₂</td>
<td>0.16</td>
</tr>
<tr>
<td>N₂</td>
<td>0.16</td>
</tr>
<tr>
<td>Gasifying agent temperature, K</td>
<td>421</td>
</tr>
<tr>
<td>Gasification pressure, MPa</td>
<td>0.42</td>
</tr>
</tbody>
</table>

3.2. The mesh

The meshes (Fig.1) used in the simulations are not the same, although of similar size and nodes distribution. The mesh created for simulation was composed of 135 000 and 102 432 mesh elements for ANSYS Fluent and CPFD Barracuda, respectively.

3.3. Solution procedure

The calculations for both models are run unsteady and the produced syngas composition at reactor outlet is monitored. After an initial period of the simulation, when a pseudo steady state is obtained, data are collected, from which mean values are calculated.

4. Preliminary results

Experimental data of gas composition at the outlet of the reactor were compared with the results of numerical simulations.

Figure 2 presents the preliminary results of obtained distribution of gas components in the gasifier.

Figure 2: Transient solid concentration and mass fraction of gas compositions.

5. Conclusions

Comparison of two numerical models (ANSYS Fluent and CPFD Barracuda) of CO₂ enhanced coal gasification in a fluidized bed reactor has been presented in the study. The three-dimensional models and simulations provide a promising way to simulate the coal gasification in fluidized beds.

References


Propagation of thermal and reaction fronts in a simple granular medium

Arkadiusz Grucelski
Institute of Fluid-Flow Machinery
Fiszera 14, 80-231 Gdańsk, Poland
e-mail: agrucelski@imp.gda.pl

Abstract

Modelling of devolatilisation phenomena is relevant for a number of industrial processes; it is also a subject of a current research interest. Despite the simultaneous developments in computing power and parallelization techniques which allow for more demanding calculations, a direct simulation of devolatilisation at the pore scale level (including a number of chemical species) is still out of reach for modern computations. Due to progress in multiscale modelling, a number of numerical tools become available for detailed simulation of phenomena occurring in a complex geometry. In the work, the modelling of coal/biomass devolatilisation at the pore scale level is attempted with the use of the Lattice Boltzmann method and the results obtained to date are presented.

Keywords: granular media, convective heat transfer, devolatilization model, Lattice Boltzmann

1. Introduction

Coking is a widely used industrial process to obtain chemically cleaner coal and coking gas. Although the process has been used for many years, a detailed description of phenomena is of recent interest of researchers. Aside of fluid flow and heat transfer in a granular medium, chemical reactions (heterogeneous as well as homogeneous) together with phenomena triggered by gas release need to be modelled at the pore level, with additional change of the pore geometry (due to change of size and shape of coal grains). The physico-chemical and geometrical complexity of these phenomena implies that more traditional tools and software of computational fluid dynamics (CFD) become prohibitively expensive as far as detailed modelling is concerned. Hence the idea of developing a multiscale approach with a microscopic (single-pore level) computation in the geometry of a representative element of volume (REV), followed by a macroscopic (system-level, unsteady 1D/2D) CFD analysis.

Devolatilisation processes, which occur with growing temperature of solid grains, are usually described as a release of light gases (like $H_2O$, $H_2$, $CH_4$, etc.) and tar from grains onto their surface (often with a simplified model of species transport inside the solid grain). Chemical phenomena occurring during heating of solid grains are well described by Solomon et al. [5]. The main focus of that work was on the detailed mechanism of species release from the coal grain. A work by Boroson et al. [1] presented a model for tar molecules breakage at high temperatures. The overall complexity of this process is briefly illustrated in referenced works; apart from presented results, those authors discuss the process from the point of view of chemical and transport phenomena.

To solve the problem of flow and heat transfer, the Lattice Boltzmann Method (LBM) has been chosen, because of its ability to deal with complex and time-varying solid-fluid interfaces, i.e. the boundaries of grains. As a first development step of the approach, the LBM has been applied to simulate fluid flow past a cylinder and in a simple granular (or porous) medium [2]. In a second step towards the physically-sound description of the process, the authors have dealt with non-isothermal flow in simple and complex geometries [3]. The present work concerns the introduction of chemical reactions (occurring at the pore level) towards a physically sound 3D simulation of the coking process in REV.

2. Numerical modelling

2.1. Model of devolatilisation

A detailed model of devolatilisation is described by Solomon in [5]; the idea presented therein is based on the approach of functional groups (FG), that constitute the grain of coal. Every FG, in the range of temperatures, releases basic gases ($CO$, $CO_2$, $H_2$, $CH_4$ etc.) and tar; the reaction rate constant is described by Arrhenius equation with coefficients presented in [5]. Additionally species are specified in terms of bond strength in the group (as loose and tight); this introduces differentiation among coefficients in the rate equation. The sum of released gases, in a time step, is next used to calculate the released tar yield from the coal grain. Tar is next a subject of homogeneous reaction, i.e., the breakage of tar molecules composed of higher hydrocarbons. This step of pyrolysis is presented in detail in [1].

2.2. Lattice Boltzmann method

The LBM is an approach developed in the early 1980s, designed originally as an extension of cellular automata to eliminate a large numerical noise. It is based on the Boltzmann equation with subsequent discretization [6]. The method has proved to be suitable for simulation of viscous and nearly incompressible fluid flows in simple and complex geometry, as well as heat transfer, with addition of chemical reactions [4]. In these cases, suitable distribution functions (for the density, internal energy or chemical species) with a variety of boundary schemes were used.

In the work the LBM variant with the flow density and velocity solved in terms of the density distribution function is used, as it is presented in [2]; the temperature field is found from the internal energy density distribution function [3]; the evolution of chemical species is governed by separate distribution functions. The form of all these LB equations is similar; an exhaustive description of LBM for fluid flow and heat transfer, together with results of 2D simulations, is available in cited works and references therein. In case of species transport, the author apply a model described in details in [4]. We develop this scheme for modelling light gases transport in 3D granular media. To this aim the D3Q15 model for fluid, heat and species transport is used. The model for species transport [4] can be briefly presented as a modified LBM where the relaxation time and equilibrium func-
tion depend on minimum density in the domain. Thanks to this modification, the model is valid even in case of locally occurring high gradients of density. For D3Q15 scheme used here, equations are well known for fluid flow and heat transfer. For the species transport, authors in [4] develop the model for D2Q5 and D2Q9 discretization schemes.

Mechanical stresses between particles (in contact) will be calculated with the Discrete Element Method (DEM), which is widely used in practical applications where there is a need to model small particles in the fluid.

2.3. Boundary schemes

Numerical modelling was performed for two geometries: a single sphere (representing a single coal grain, Fig. 1) and a random granular medium (representing a bed of coal grains). Numerical domain is discretized with use of $2N \times N \times N$ nodes where $N = 60$. The diameter of sphere is $d = 0.25N$ whereas in the granular medium $d$ is the mean grain diameter and the standard deviation is $\delta d = 0.05N$. In both cases boundary conditions are the same. At the longer sides of the numerical domain (parallel to the main flow direction) fully periodic conditions is applied. At the boundaries perpendicular to flow direction, in case of fluid flow the author use the shift–periodic boundary conditions (with calculated pressure drop between inlet and outlet); details are described in [2]. For heat transfer a prescribed inlet temperature and the equilibrium distribution function is applied; at the outlet the temperature is extrapolated. For the time being, simple periodic boundary conditions at inlet/outlet for chemical species is used.

At the solid grain surface, the non-equilibrium boundary scheme is applied for fluid flow and heat transfer: for fluid flow the no-slip condition is used. Additionally to improve the stability of heat transfer computations, a local averaging of the temperature is performed at the grain surface. Chemical species release from grains is modelled with the use of mass flux at the surface.

3. Benchmarks

3.1. Single grain of coal

The first chosen benchmark was the devolatilisation of a single spherical grain. This case was used to compare the global mass of chemical species in the solid grain (trapped in the functional group), in the fluid (already released to the fluid or at the grain surface), and (not presented here) the mass of species already left through the outlet boundary.

Figure 1 presents the LBM results of non-dimensional global mass for a chosen species in whole domain (the dotted line on the bottom plot); the same plot also illustrates the overall mass flux on the surface and the averaged temperature of the solid grain. Double maximum value in flux value is connected with strength of bonds of the species to coal molecules. Mass fluctuation (not visible here) is due to a numerical artefacts and is typical of LBM.

3.2. Random layout of spherical grains

For the modelling of devolatilisation processes in granular media, a random layout of the obstacles (Fig. 2) is used. Additional simulations for obtaining yields of gaseous products for comparison with results with reference data (see [5]) are in progress.

4. Concluding remarks

Detailed simulation of coking is still under development. The presented results partially cover the breakage of tar molecules. An exhaustive test case is under preparation for quantitative check of species mass in the system and the total energy. After introducing homogeneous reactions for tar, more effort will be put on construction of a 3 species model (coal $\rightarrow$ tar + gases + coke), to be compared with existing models known from literature.

References


Dynamics of heat and mass transfer in a moving reactive granular bed

Dariusz Kardaś1, Sylwia Polesek-Karczewska2, Przemysław Cizmiński3

1,2,3 Department of Renewable Energy, Institute of Fluid Flow Machinery, Polish Academy of Sciences
Fiszera 14, 80-231 Gdańsk, Poland

e-mail: dk@imp.gda.pl1, sylwia.polesek-karczewska@imp.gda.pl2, pcizmiski@imp.gda.pl3

3 Gdańsk University of Technology, Poland
Narutowicza 11/12, 80-233 Gdańsk, Poland

Abstract

The paper focuses on the mathematical description of thermal and flow processes in a biomass downdraft gasifier. The analysis is based on transient one-dimensional model includes the problems of heat transfer, water evaporation and condensation, and devolatilization, and is limited to the upper part of the reactor. Special emphasis was put on the speed of bed movement, which is of practical interest in improving the gasification technology. To solve the problem that accounts for particle settling an Eulerian-Lagrangian method was employed. Obtained results indicated on the impact of the settling speed on the extension of drying and pyrolysis zones.

Keywords: gasifier, biomass particles, Eulerian-Lagrangian approach, bed settling

1. Introduction

The phenomena of heat and mass transfer in a reactive granular bed are of general importance in the problems associated with thermochemical conversion of solid fuels, including for instance the gasification technology. Such process is intended to generate the flammable gas, which may serve to run the modules for electricity production, e.g., turbines, engines. The process effectiveness, and hence the gas yield and quality, are strongly dependent on a large number of factors, in particular on the temperature distribution in a reactor, chemical composition and size of fuel particles and gasifying agent. Selecting optimal operating parameters requires the knowledge about the physical and chemical phenomena occurring inside the gasifier, which involves the problems of heat transfer, phase transitions due to evaporation/condensation of moisture and tars, transport of evolved gases and vapor, as well as chemical reactions.

Many efforts have been focused on the mathematical modeling of gasification to provide reliable predictions of the process dynamics. The majority of the models proposed in literature relates to the steady state and are devoted to various aspects of the technology. The results of numerical studies regarding temperature distribution and gas composition in a downdraft gasifier were presented for example in Ref. [1, 2, 3]. However, there is a lack of models that would take into consideration the speed of bed movement which results from the bed settling as the process is in progress and the particle volume decreases. The dynamics of bed settling affects the layout of particular zones in a reactor, and consequently the course of the total process.

The work aims at the analysis of the dynamics of heat and mass transfer in a gasifier when taking into account the particle bed settling. The one-dimensional non-stationary model was proposed. In order to determine the speed of bed movement, the Lagrangian approach was used.

2. Modeling of gasification process

Four zones appear to be recognized in a downdraft gasifier: (i) drying where the moisture evaporates, (ii) pyrolysis in which the dry fuel decomposes and the gas is released, (iii) combustion of devolatilized fuel particles and finally, (iv) the reduction where the surface reactions between the char and gas take place. The analysis reported herein is limited to the upper part of a gasifier, situated right above the oxidation zone, where the drying and pyrolysis take place, as it is shown in Fig. 1.

![Figure 1: A draft of the studied downdraft gasifier](image-url)

2.1. Governing balance equations

The basic set of equations consists of mass balance equations for biomass particles (b), moisture (w), vapor and gas mixture (g), momentum balance equation for gaseous phase and energy balance equation for wet biomass bed, as follows respectively

\[
\frac{\partial (\varepsilon_b \rho_b v)}{\partial t} + \frac{\partial (\varepsilon_b \rho_b v)}{\partial z} = -\dot{W}_{bg},
\]  

where \(\dot{W}_{bg}\) is the rate of biomass flux.
\[
\frac{\partial (\varepsilon_w \varrho_w)}{\partial t} + \frac{\partial (\varepsilon_w \varrho_w v)}{\partial z} = -\dot{W}_{wg}\tag{2}
\]
\[
\frac{\partial (\varepsilon_w Y_w \varrho_g \vartheta_g)}{\partial t} + \frac{\partial (\varepsilon_w Y_w \varrho_g v \vartheta_g)}{\partial z} = \dot{W}_{wg}\tag{3}
\]
\[
\frac{\partial (\varepsilon_g \varrho_g \vartheta_g)}{\partial t} + \frac{\partial (\varepsilon_g \varrho_g v \vartheta_g)}{\partial z} = \dot{W}_{bg} + \dot{W}_{wg}\tag{4}
\]
\[
\frac{\partial (\varepsilon_g \varrho_g v^2)}{\partial t} + \frac{\partial (\varepsilon_g \varrho_g v^2)}{\partial z} =
\]
\[
- \frac{\partial (\varepsilon_g \varrho_g)}{\partial z} - \frac{\mu_g v \vartheta_g}{K} + \mu_g \frac{\partial^2 v \vartheta_g}{\partial z^2}\tag{5}
\]
\[
\frac{\partial (\varepsilon_g \varrho_g \vartheta_g + \varepsilon_w \varrho_w u_w)}{\partial t} + \frac{\partial (\varepsilon_g \varrho_g \vartheta_g + \varepsilon_w \varrho_w u_w)}{\partial z} =
\]
\[
\frac{\partial}{\partial z} \left( \lambda \frac{\partial T}{\partial z} \right) - \dot{W}_{kg} h_{bg} - \dot{W}_{wg} h_{wg}\tag{6}
\]

In the given equation system \( \varepsilon \) denotes volume fraction, \( \varrho \) [kg/m³] density, \( v \) [m/s] speed of bed settling, \( \varrho_g \) [m/s] is the flow velocity of released gas, \( T \) [K] temperature, \( u \) [J/kg] specific internal energy, \( h_{bg} \) and \( h_{wg} \) [J/kg] are the enthalpies of pyrolysis and vaporization, respectively, whereas \( \lambda \) [W/(m K)] is the effective thermal conductivity of the wet biomass. Terms \( \dot{W}_{bg} \) and \( \dot{W}_{wg} \) [kg/(m²s)] are the mass rates for gas and vapor, respectively, described in Ref. [4, 5]. Quantity \( Y_{w,g} \) represents the mass fraction of water steam in gas mixture and \( K \) [m²] the permeability.

Mass balance equations, Eqns (1) – (4), were solved using explicit numerical schemes, whereas to determine temperature and pressure the implicit method was applied.

2.2. Bed movement speed

In order to estimate the dynamics of the bed movement, the Lagrangian approach was employed. A group of particles lying one above another, on the height between the oxidation zone and the highest level of the feedstock bed, are considered. It is assumed that particles are spherical and initially of the uniform size. The reduction in their volume results from devolatilization and vaporization, respectively, whereas the temperature was considered to change linearly from ambient to 800 K. The initial speed at the level of oxidation zone was 20 m/s. Figure 2 illustrates the change in volume of particles for three time instants. A schematic outline of the particle sequence for the case of \( t = 2 \) h was also given (top of the figure).

As observed, at this time only the particles situated between the levels of 0.06 and 0.11 m, which is the temperature area corresponding with an intensive pyrolysis, change their volume.

4. Remarks

The proposed one-dimensional transient model of heat and mass transfer in a moving bed based on Eulerian-Lagrangian method allows to predict the distribution of temperature, pressure, gas and moisture evolution in a settling bed. The change dynamics of these parameters and the speed of bed movement was determined. The latter, as the results showed, affects the layout of the studied pyrolysis and drying zones in a gasifier.

References

The effect of granular temperature formulation in the two-fluid model for a turbulent fluidization of glass beads

Adam Klimanek1,2*, Wojciech Adamczyk2, Gabriel Węcel3, Andrzej Szlęk4

1,2,3,4 Institute of Thermal Technology, Silesian University of Technology
Konarskiego 22, 44-100 Gliwice, Poland

e-mail: adam.klimanek@polsl.pl1, wojciech.adamczyk@polsl.pl2, gabriel.wecel@polsl.pl3, andrzej.szlek@polsl.pl@polsl.pl4

Abstract

In this paper the effect of applying the full partial differential equation for granular temperature and its simplified form in a turbulent fluidization regime was examined. Four cases were studied in which particles of two different mean diameters were fluidized at two fluidization velocities. Mean solids volume fractions and velocities were compared. The results showed that applying the simplified and more stable algebraic equation has relatively small effect on the predicted flow fields, but near the walls, where the solids volume fractions were smaller and falling velocities higher.

Keywords: Eulerian-Eulerian, fluidized bed, kinetic theory of granular flow, granular temperature, turbulent fluidization regime

1. Introduction

Fluidized Beds (FBs) are frequently used in industry for realization of variety of processes. Modelling of the FBs by means of CFD based models is challenging due to complexity of the multiphase flows. This is associated with large scale differences between the particle level and geometry of the equipment. The scale difference is also a reason of long computational times of these transient flows. One of the approaches used to model gas-particle flows is the Eulerian-Eulerian (two-fluid or multi-fluid) approach in which both the solid and the fluid phase are treated as interpenetrating continua. The solid phase is represented by its density and a single characteristic diameter. The Eulerian-Eulerian approach is a well-established model, frequently applied to gas-solid flow and specifically to circulating fluidized beds modelling of small and large industrial systems, just to mention a few. In the Eulerian-Eulerian approach many closure models are required to account for the fluid-solids as well as solids-solids momentum transfer. For the former, drag correlations are used [1,2]. For the latter, use is made of the Kinetic Theory of Granular Flow (KTGF) [1] which allows determination of granular temperature, solids pressure and solids shear stresses. If the closure terms are determined the governing equations of the Eulerian-Eulerian model can be solved. As a result volume fraction, velocities and pressure of the solid and fluid phase distribution in the flow domain are determined. The granular temperature can be determined from a transport equation in which the convection and diffusion terms are neglected. Therefore a simplified version of Eq. (1) has been developed [2] in which the convection and diffusion terms are neglected.

3. The analysed cases

The effect of simplification of the granular temperature equation was examined in simulations of a fluidized bed operated in a turbulent fluidization regime. The fluidized materials were glass beads of density \( \rho_s = 2478 \text{ kg/m}^3 \) and two diameters \( d_p = 281 \mu\text{m} \) and \( d_p = 548 \mu\text{m} \). For each diameter two air fluidization velocities were examined. In Table 1 the input data used in the simulations are summarized. For each case the fluidized bed was simulated using the partial differential equation (1) and the simplified algebraic equation. To distinguish the cases the names pdeg and aegt were used, respectively.

4. The numerical model

The calculations have been done using the ANSYS Fluent 14.0 software.

4.1. Geometry and mesh

Geometry of the model encompasses a 3 m high and 0.6 m wide fluidized bed represented in two dimensions as shown in Figure 1. The gas is introduced through the bottom wall. A velocity inlet boundary condition is used.

This study has been supported by the statutory research fund of the Silesian University of Technology, Faculty of Energy and Environmental Engineering, Institute of Thermal Technology.
Table 1: Main model input data

<table>
<thead>
<tr>
<th>Analysed case</th>
<th>c1</th>
<th>c2</th>
<th>c3</th>
<th>c4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluidization velocity, m/s</td>
<td>1.61</td>
<td>3.20</td>
<td>3.29</td>
<td>5.77</td>
</tr>
<tr>
<td>Gas density, kg/m³</td>
<td>1.21</td>
<td>1.16</td>
<td>1.14</td>
<td>1.09</td>
</tr>
<tr>
<td>Particle diameter, µm</td>
<td>281</td>
<td></td>
<td>548</td>
<td></td>
</tr>
<tr>
<td>Outlet pressure, kPa</td>
<td>99.8</td>
<td></td>
<td>101.2</td>
<td></td>
</tr>
<tr>
<td>Particle density, kg/m³</td>
<td></td>
<td>2478</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Packing limit</td>
<td></td>
<td>0.62</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The fluidized material and the gas can leave the bed through the top boundary where constant static pressure is applied defined in Tab. 1. The particles that leave the bed are recirculated through the 0.2 m return leg shown in Fig. 1 so that a constant mass of solid phase is retained in the fluidized bed. This is realized by a user defined function. The mesh was built out of square cells in the rectangular section and quad cells in the return leg. For the smaller particles the square mesh elements were of 3 mm side and for the larger particles of 6 mm side.

Figure 1: Geometry of the model

4.2. Solution procedure

The calculations are run unsteady and data are collected during the calculations for 20 s. The collected data are used to compute mean values and the mean values are compared.

5. Results

Calculated mean solids volume fraction distribution along the width of the fluidized bed at 0.25 m and 1.3 m above the air inlet for cases c1 and c2 (Fig. 2) and cases c3 and c4 (Fig. 3) are presented. As can be seen the results are comparable at both heights. Slightly lower mean solids volume fractions are obtained for the partial differential equation (pdeg) specifically at higher elevations were the neglected terms in Eqn (1) become more important. Substantial differences are observed near the wall. The predicted solids volume fraction near the walls are much lower when the simplified equation is used (aegt).

Figure 2: Mean solids volume fraction distribution along the width of the fluidized bed at 0.25 m and 1.3 m above the air inlet for cases c1 and c2

Figure 3: Mean solids volume fraction distribution along the width of the fluidized bed at 0.25 m and 1.3 m above the air inlet for cases c3 and c4

6. Conclusions

Comparison of the results obtained for turbulent fluidization regime showed that application of the full partial differential equation for granular temperature and the simplified form produces comparable results but near the walls where considerably smaller values were obtained. Similar behaviour was observed for the solids velocities, however the vertical velocities near the wall were higher for the simplified equation.

References


Effect of biomass settling in the fixed bed gasification reactor

Jacek Kluska

Institute of Fluid-Flow Machinery, Polish Academy of Sciences
14 Fiszera St, 80-231 Gdańsk
e-mail: jkluska@imp.gda.pl

Abstract

The paper presents the problem of fuel settling during gasification process, which means the movement of a solid phase in a fixed bed reactor, that was used to describe the motion of the biomass particles as a result of change in their volume. The loss of volume is caused by pyrolysis and chemical reactions. This work shows the elementary and technical analysis of pine wood and experimental results of fuel settling velocity in the small scale gasification reactor. Research was carried out in a reactor operating in a batch mode for different amounts of air supplied to the gasifier. The results show that the settling fuel velocity is varies with the height of the reactor as well as the velocity depends on the amount of supplied air to the gasification process.

Keywords: gasification, settling velocity, pine wood, fixed bed, pyrolysis

1. Introduction

During the gasification process in a fixed bed reactor operating in a batch mode, fuel is delivered from the top and moves down. Typical processing of biomass in a fixed bed gasifier starts from drying, where moisture is released from biomass, then followed by pyrolysis in which gases and char are produced from biomass [1]. In the next phase of gasification process gases are mixed with air and combusted to CO2 and H2O, whereas charcoal combines with CO2 and H2O forming a combustible gas. The most important factors in the gasification process are the temperature and the amount of air supplied to the gasification process [2,3]. These parameters have a significant impact on the quality of the produced syngas. However, in terms of entire process of gasification important role plays fuel particle velocity. This work presents experimental investigation of fuel settling velocity during gasification of pine wood in the small scale batch reactor.

2. Wood sample

In order to determine the properties of the tested biomass, pine wood was minced first in a knife mill and then in a centrifugal mill equipped with a sieve with the mesh diameter of 0.2 mm. The results of the technical and elementary analysis are shown in the Tab. 1.

Table 1: Technical and elementary analysis of the biomass

<table>
<thead>
<tr>
<th>LHV [MJ/kg]</th>
<th>19.59</th>
</tr>
</thead>
<tbody>
<tr>
<td>Technical analysis [%]</td>
<td></td>
</tr>
<tr>
<td>Moisture content</td>
<td>8.4</td>
</tr>
<tr>
<td>Volatile</td>
<td>67.9</td>
</tr>
<tr>
<td>Char</td>
<td>21.4</td>
</tr>
<tr>
<td>Ash</td>
<td>2.3</td>
</tr>
<tr>
<td>Elementary analysis [%]</td>
<td></td>
</tr>
<tr>
<td>C</td>
<td>45</td>
</tr>
<tr>
<td>H</td>
<td>6.4</td>
</tr>
<tr>
<td>O</td>
<td>47.3</td>
</tr>
<tr>
<td>N</td>
<td>1.3</td>
</tr>
</tbody>
</table>

3. Experimental stand

Experimental studies were carried out in a small scale reactor with a height of 100 cm and the 22 cm diameter (Fig. 1). For the purposes of the research reactor was operated in batch mode.

Figure 1: Small scale gasification reactor

The hot reactor was filled with char up to air nozzles, which are located at a distance of 45 cm from the top of the reactor. Next the reactor was filled up with wood chips (2.8 kg) and closed together with the measurement piston. The tests were performed for pine wood chips of 3 - 5 mm and a moisture content of 15%. Experiments were carried out for 5 different amounts of air supplied to the gasification process: 6.65, 11.07, 13.29, 15.51 and 19.94 Nm³/h [4]. In each experiment time of piston settling on a distance about 5 cm was measured (Fig. 2). The reactor was filled up again with fresh fuel after settling of the piston with the fuel to the level of air nozzles.
4. Results

The Table 2 summarizes the fuel settling time for particular experiments [4]. The results show that the shortest settling time of fuel occur at a height between 15 and 25 cm.

Table 1: The time of fuel settling in the gasifier for different air flows

<table>
<thead>
<tr>
<th>[Nm3/h]</th>
<th>6.65</th>
<th>11.1</th>
<th>13.29</th>
<th>15.51</th>
<th>19.94</th>
</tr>
</thead>
<tbody>
<tr>
<td>40</td>
<td>340</td>
<td>197</td>
<td>425</td>
<td>199</td>
<td>210</td>
</tr>
<tr>
<td>35</td>
<td>358</td>
<td>175</td>
<td>145</td>
<td>141</td>
<td>80</td>
</tr>
<tr>
<td>30</td>
<td>140</td>
<td>90</td>
<td>75</td>
<td>53</td>
<td>30</td>
</tr>
<tr>
<td>25</td>
<td>120</td>
<td>60</td>
<td>45</td>
<td>35</td>
<td>26</td>
</tr>
<tr>
<td>20</td>
<td>70</td>
<td>112</td>
<td>25</td>
<td>21</td>
<td>18</td>
</tr>
<tr>
<td>15</td>
<td>200</td>
<td>250</td>
<td>15</td>
<td>54</td>
<td>30</td>
</tr>
<tr>
<td>10</td>
<td>310</td>
<td>320</td>
<td>40</td>
<td>70</td>
<td>35</td>
</tr>
<tr>
<td>5</td>
<td>370</td>
<td>340</td>
<td>480</td>
<td>130</td>
<td>104</td>
</tr>
</tbody>
</table>

Table 2: Fuel consumption during gasification of birch chips for various air flows

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Biomass consumption [kg/h]</td>
<td>4.74</td>
<td>5.86</td>
<td>7.88</td>
<td>12.89</td>
<td>17</td>
</tr>
</tbody>
</table>

5. Conclusion

Experimental work was carried out in the small scale reactor operating in batch mode. Tests of fuel settling were performed from the top of the reactor to the level of air nozzles for different amount of air supply to the gasification process. The results show that the settling fuel velocity changes with the height of the reactor. The highest rate settling fuel was observed below zones of pyrolysis and above volatile combustion zone. It is associated with decrease of volume of fuel particles, which is caused by high temperature.

The experimental results indicate that the rate of biomass gasification process depends on the temperature and the amount of supplied air to the gasifier. These parameters have a direct impact on the rate of consumption of biomass and the velocity of the fuel settling in the reactor.

References

Study on the impact of char properties on the gasification effectiveness

Sylwia Polesek-Karczewska¹, Mateusz Szumowski²

¹,² Department of Renewable Energy, Institute of Fluid-Flow Machinery, Polish Academy of Sciences
Fiszera14, 80-231 Gdańsk, Poland

e-mail: sylwia.polesek-karczewska@imp.gda.pl, matszumowski@gmail.com

Abstract

The paper reports the study on the impact of sample size on the properties of char formed during high-temperature pyrolysis and its influence on the gasification effectiveness. Pine wood particles were pyrolyzed at temperature up to 1000°C. The sample losses of mass and volume, apparent density and changes in surface area of char were studied. The next step of investigation carried out was e gasification of the obtained char samples employing various process parameters.

Keywords: pyrolysis, gasification, woody biomass, size of sample, process conditions

1. Introduction

Growing requirements for environmental protection motivate researchers and entrepreneurs to develop alternative methods of energy production based on renewable raw materials such as woody biomass. Gasification and pyrolysis are, next to burning, the fundamental technologies of thermo-chemical conversion of solid fuels. Gasification defined as the number of sub-processes taking place in the gasification reactor which covers the processes of: drying, pyrolysis, combustion and finally, reduction in a bed of char. Unlike the combustion, the only purpose is to produce heat, gasification aims at generating the combustible synthesis gas which consists mainly of hydrogen, carbon monoxide and methane. The produced gas thus represents the second-generation fuel that may be used in power production systems. The key aspects of syngas utilization is its purity and composition. The latter depends to large extent on the bed of char (e.g. grain texture, packing density), formed while successfully passing through pyrolysis and oxidation zones. These gas features, in both quantitative and qualitative terms, may be controlled by the process factors and the characteristic of the feedstock. From this point of view, the experimental analyses of the influence of basic parameters on the process yields are of major practical importance.

In order to recognise the effect of char bed characteristics on gasification, i.e., basically the grain size and structure, the two-step experimental investigation was performed. The biomass particles were first pyrolyzed in order to obtain chars of various properties. This test procedure is then followed by char gasification in other reactor.

2. General process characteristics

The chemistry of gasification is described by homo- and heterogeneous reactions. Homogenous reactions are secondary reactions between gases. The most important are heterogeneous reactions between a solid phase and gaseous components which determine the feedstock conversion. The main heterogeneous reactions occurring during the gasification process like Boudouard’s reaction (R1), hydrogenation (R2), combustion (R3), reforming (R4) and methanation (R5) are given as follows [1].

\[
\begin{align*}
C + CO_2 & \rightarrow 2CO \quad \Delta H = +172 \text{ MJ/kgmol} \quad (R1) \\
C + H_2O & \rightarrow CO + H_2 \quad \Delta H = +131 \text{ MJ/kgmol} \quad (R2) \\
C + O_2 & \rightarrow CO_2 \quad \Delta H = -394 \text{ MJ/kgmol} \quad (R3) \\
C + H_2O & \rightarrow CO_2 + 2H_2 \quad \Delta H = +88 \text{ MJ/kgmol} \quad (R4) \\
C + H_2 & \rightarrow CH_4 \quad \Delta H = -74,8 \text{ MJ/kgmol} \quad (R5)
\end{align*}
\]

The effectiveness of heterogeneous reactions depends on many process factors like temperature and total pressure but also for the characteristic of a char. Char is a solid residue consisting mainly of elementary carbon, formed in a pyrolysis process which take place under non-oxygen conditions. The char properties are influenced by many variables such as the characteristic of virgin biomass and pyrolysis conditions like temperature, heating rate or total pressure [2].

The study the influence is checked of the size of wood particles on a high-temperature pyrolysis course and the characteristic of formed char. The main parameters to be checked are the change of sample mass (conversion), porosity and volume. The conversion degree of feedstock can be calculated as a weight loss according to:

\[
X = \frac{m_0 - m}{m_0}
\]

where \(m_0\) and \(m\) denote the initial and final weight of the sample, respectively.
3. **Experimental procedure**

Experiments were carried out in a high-temperature reactor with induction heating which allows to carry out the processes of thermal decomposition of the raw materials under isothermal conditions at temperatures up to 1000°C (Fig. 1). The reactor chamber is made of stainless steel type H23N18 MIG welded technique using high alloy steel 307 Si characterized by corrosion and mechanical stress resistance. The reactor chamber is equipped with two thermocouples capable of recording temperatures of 1/3 and 2/3 of the height of the reactor. The installation is designed for carrying out the gasification and pyrolysis processes of biomass, agricultural waste, municipal waste, RDF and others.

In the study the samples from pine wood with different length to width ratio were used. The conversion and apparent density of feedstock, as well as the calorific value of the formed char were determined. In order to do so the elemental and technical analyses of samples before and after experiment were performed.

In order to provide non-oxygen conditions the reactor chamber was flushed with an inert gas – nitrogen.

The experiments made are the next step of conducted research on low-temperature pyrolysis of wood biomass. The preliminary results of loss in mass ($\Delta m$) and volume ($\Delta V$) of pine wood cylinders pyrolyzed at 270, 320, 370, 420, 470 and 520°C (Fig. 2) are summarized in Tab. 1. As shown the feedstock conversion grows with temperature increase except the case for 370°C. This tendency is caused by the endothermic character of chemistry reactions of pyrolysis. It should be underlined that the results regarding volume change for temperatures 420 and 520°C seem to be questionable. Therefore, the measurements will be repeated, also for higher temperatures up to 1000°C.

![Figure 1: Reactor for high-temperature conversion of solid fuels](image)

![Figure 2: Initial samples of pine wood and char formed in the pyrolysis at 240, 270, 320, 370, 420, 470, 520°C [3]](image)

<table>
<thead>
<tr>
<th>$T$ [°C]</th>
<th>$\Delta m$ [%]</th>
<th>$\Delta V$ [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>240</td>
<td>23.54</td>
<td>13.33</td>
</tr>
<tr>
<td>270</td>
<td>45.61</td>
<td>32.67</td>
</tr>
<tr>
<td>320</td>
<td>63.86</td>
<td>50.83</td>
</tr>
<tr>
<td>370</td>
<td>71.23</td>
<td>63.00</td>
</tr>
<tr>
<td>420</td>
<td>69.58</td>
<td>34.17</td>
</tr>
<tr>
<td>470</td>
<td>69.72</td>
<td>59.33</td>
</tr>
<tr>
<td>520</td>
<td>74.39</td>
<td>79.83</td>
</tr>
</tbody>
</table>

4. **Summary**

The paper is focused on the analysis of an impact of the wood particle size on the properties of char formed in a high-temperature pyrolysis considered one of the factors determining the effectiveness of biomass gasification. The obtained chars will be further used in a high-temperature gasification under different process conditions, such as type and flow rate of gasifying agent, to compare the course of process for various char beds being defined by particle size and its apparent density. Optimization of wood chips will allow to get homogenous char, to be efficiently gasified.

**References**


The closures problem in the CFD area

Wojciech Sobieski
Faculty of Technical Sciences, University of Warmia and Mazury in Olsztyn
Oczapowskiego 11, 10-957 Olsztyn, Poland
e-mail: wojciech.sobieski@uwm.edu.pl

Abstract

In the article, the problem of closures of the main balance equations in the CFD area is shown. As a background to the discussion, a numerical model of an existing spouted bed grain dryer was used. To obtain a virtual model, consistent with the experiment on the level of dynamics, six different unit issues had to be mathematically described. Since each of these has a few options, the matrix size of possibilities grows significantly with each new theory. In the article, solution to this problem is proposed, using the so-called basic model. The motivation for writing the article stems from the author’s experience in numerical modelling of different systems, especially multiphase flows. The objective of the article is to emphasize the importance of the selection of closures in numerical modelling.

Keywords: CFD, closures, Eulerian Multiphase Model, fluidization

1. Introduction

Currently the term Computational Fluid Mechanics (CFD) covers a wide range of different numerical methods. One of them is the Finite Volume Method, which allows creating numerical simulations of one phase or multiphase flows (with the use of Eulerian or Lagrangian approach). All of these methods are usually created on the base of main balance equations of mass, momentum and energy and, in addition, different supplementary equations, the so-called closures, i.e. specific models describing particular issues. The number of closures is sometimes huge. Practically every issue is attached many solution propositions to be found in the literature. This situation occurs in every research area. The multiplicity of possibilities results in two main questions [7]: what is the available set of solutions for considered problem, and which of them should be used in the numerical model of the moment? The situation becomes complicated when the topical numerical model contains not one, but many locations where an appropriate closure must be chosen.

In the article, an example of creation of the numerical model of dynamics in a spouted bed grain dryer is shown. The experiment, as well as the so-called basic model, are described in the article [4]. The other aspects of using the EMM in the context of spouted fluidization were shown in articles [1, 2, 3, 5, 6].

2. General numerical model

Spouted fluidization is usually modelled with the use of the EMM. In the literature, it is sometimes referred to as the two-fluid or multi-fluid model. The EMM is a highly generalized model for describing mixtures of any number of phases: gas, fluid, and solid particles. A separate system of mass, momentum, and energy balance equations is solved for each phase. Phases are coupled through pressure and interphase mass, momentum, as well as energy exchange coefficients. Those coefficients are characteristic for the model, playing a key role. A description of interactions between different phases is largely dependent on a question if the flow concerns liquid phases only or liquid as well as solid ones. All phases are described in the Eulerian approach. General balance equations of mass (1), momentum (2), and energy (3) take the following form in EMM [4, 5]

\[
\frac{\partial}{\partial t}(\alpha_q \rho_q) + \text{div}(\alpha_q \rho_q \vec{v}_q) = M + S_{m,q} \tag{1}
\]

\[
\frac{\partial}{\partial t}(\alpha_q \rho_q \vec{v}_q) + \text{div}(\alpha_q \rho_q \vec{v}_q \vec{v}_q) = \text{div}(\vec{\tau}_q - (\alpha_q \rho_q \vec{v}_q \vec{v}_q) + \text{div}(\vec{\tau}_q \vec{v}_q + \vec{\rho}_q) + Q + S_{h,q} \tag{2}
\]

\[
\frac{\partial}{\partial t}(\alpha_q \rho_q h_q) + \text{div}(\alpha_q \rho_q h_q \vec{v}_q) = \text{div}(\vec{\tau}_q \vec{v}_q + \vec{\rho}_q) + Q + S_{h,q} \tag{3}
\]

where \( \alpha_q \) is the volume fraction \([-] \), \( \rho_q \) is the density \([kg/m^3]\), \( \vec{v}_q \) is the velocity \([m/s]\), \( M \) is the mass exchange coefficient between phases \([kg/(m^3 s)]\), \( S_{m,q} \) is the additional mass source \([kg/(m^3 s)]\), \( \vec{\tau}_q \) is the total stress tensor \([Pa]\), \( p \) is the static pressure of the mixture \([Pa]\), \( \vec{R} \) is the solid pressure (a correction factor for fluid-solid systems) \([Pa]\), \( \vec{I} \) is the unit tensor \([-] \), \( \vec{\tau} \) is the momentum exchange coefficient between phases in motion \([N/m^3]\), \( S_{F,q} \) is the additional source forces impacting the phase \([N/m^3]\), \( h_q \) is the enthalpy \([J/kg]\), \( \vec{q}_q \) is the heat flux \([J/(m^2 s)]\), \( Q \) is the energy exchange coefficient between phases \([J/(m^3 s)]\) and \( S_{h,q} \) is the additional heat source \([J/(m^3 s)]\). The lower index denotes the \( q \)-th component of the mixture.

3. The example of the closures problem

The closure problem may be illustrated on the example of momentum balance equation. The issues that must be additionally defined are collected in Tab. 1. Only in this example, 80640 combinations of closures may be created. Constantly new ideas are published and so this number grows every year. Of course, it is not possible to test all these combinations. The best way is to find the most popular models and choose the one, which may be considered the basic model. Here it was the so-called Gidaspow model (this is the most widely used model in this area). In the next stage, every specific issue may be investigated separately (like in the cycle of publications cited in this work). In this way, the impact (importance) of every individual issue becomes recognized.
Table 1: Sample EMM closures applied in numerical modeling of spouted beds

<table>
<thead>
<tr>
<th>No</th>
<th>The issue</th>
<th>The closure</th>
<th>Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>Solid stress tensor [3]</td>
<td>Johnson and Jackson, Lun et al., Syamlal et al., Ocone et al., Gidaspow, Hrenya and Sinclair, Laux</td>
<td>7</td>
</tr>
<tr>
<td>3</td>
<td>Granular pressure [3]</td>
<td>Lun et al., Ma-Ahmad, Ding and Gidaspow, Syamlal-O’Brien</td>
<td>4</td>
</tr>
<tr>
<td>4</td>
<td>Drag coefficient [1]</td>
<td>Stokes, Oseen, “classic”, Schiller and Neumann, Dallavalle, Clift and Gauvin, Ihme, Clift et al., Concha and Barrientos, Cheremisinoff and Gupta, Flemmer and Banks, Turton and Levenspiel, Khan and Richardson, Orzechowski and Prywer, Umbel, Brown and Lawler, Almedei, Cheng</td>
<td>18</td>
</tr>
<tr>
<td>5</td>
<td>Switch function (in Gidaspow model) [6]</td>
<td>based on atan(x) function, based on exp(x) function</td>
<td>2</td>
</tr>
</tbody>
</table>

In Fig. 1, the volume fraction of the granular phase in the spout axis is shown. During investigations, it turned out that in addition to the Gidaspow model there might be distinguished four other sets of closures giving results on the same levels of quality and quantity. Other combinations of closures gave results inconsistent with the experiment. These results show that the search for other methods of modelling (sets of closures) is justified, as there is a possibility that one of these ways gives a better compatibility with the experiment.

![Volume fraction in dryer axis](image)

Figure 1: Examples of modeling the same spouted bed using several closure sets

It should be mentioned that not only closures are important while creating a numerical model. Other aspects are also important [5], like for example the number of dimensions (2D, 3D), selection of the computing area, the kind and quality of the grid, approach to the turbulence modelling and many others. They should be tested in the first stage, before the basic model is selected. In other case, it can happen that at a later stage of research (e.g. after publishing results), a critical error will be discovered and the whole effort will be crossed out.

4. Summary

The following conclusions can be formulated based on the results of the study:

- The large number of different theories and formulas leads to difficulties in choosing appropriate closures of the main system of balance equations.
- The process of creation of each numerical model must be preceded by a detailed analysis of the literature. In many cases, the knowledge is "fuzzy" and the author must organize this knowledge before continuing research. Sometimes reference to the original sources is required.
- In many cases, more than one set of closures gives acceptable results.
- In many cases, none of the available models gives satisfactory results.
- It is recommended to check different sets of closures in every case. It makes no sense to test all possible combinations: the better solution is to define the basic model, and next to investigate the impact of every factor separately.
- If the closure introduces a numerical value, it is recommended to perform a sensitivity analysis. In this case, the order of the importance of the particular closures (laws, formulas, etc.) may be obtained.

References

The course of products concentration of plasma pyrolysis of rubber in function of plasmatron power

Jarosław Szuszkiewicz
Faculty of Technical Sciences, University of Warmia and Mazury in Olsztyn
ul. Oczapowskiego 11, 10-719 Olsztyn, Poland
e-mail: jerry@uwm.edu.pl

Abstract

The problem of rubber waste disposal has not been solved entirely yet. There have been many methods worked out but thermal ones seem to be the most promising considering energy potential lurking in rubber. Depending on the composition of rubber material it might consist over 90 % of C and H₂. In order to utilize rubber the plasma pyrolysis method was applied. The main purpose of the paper is to investigate the influence of plasmatron power and plasma gas composition on the production of gaseous products.

Keywords: plasma pyrolysis, rubber waste, utilization, plasmatron power

1. Introduction

Rubber waste, mainly used tires, is an environmental issue in Poland and all over the world. Much effort has been taken to solve this problem [4,5]. Many methods for rubber recycling were worked out but none of them was fully successful and profitable.

It seems that thermal utilization methods of rubber are the most promising ones. Thermal processing is capable of 100 % decomposition of rubber to gas and liquid mainly. The temperature level usually yields a dominant phase: the higher temperature the bigger amount of gas products [2]. Only thermal decomposition of rubber is the only means to generate energy out of it.

The used tires cover 75 % of rubber waste. In Poland used tires comprise over 150 thousand tons annually [3]. Thermal methods which have been applied so far are not satisfactory. They produce heat by means of traditional incineration. They are not efficient because of huge heat loses. To make matters worse, a lot of dangerous chemical compounds is emitted to the air during processing.

A thermal method using a heat source different than fire would be more adequate. The pyrolysis decomposition does not need oxygen to operate. If one uses plasma as the heat source in pyrolysis then application of high temperatures (over 6 thousand K) is capable of decomposing almost every material, including rubber, avoiding synthesis of many harmful compounds. So, plasma pyrolysis seems to be the right choice for rubber waste utilization.

2. Research test-stand

The research on plasma pyrolysis of rubber waste has been carried out in the Faculty of Technical Sciences of the University of Warmia and Mazury in Olsztyn. The diagram of the test-stand is presented in Fig. 1. It consists of a DC plasma generator (plasmatron), power supply unit, reactor, cooling systems of the plasmatron and reactor, fluidal feeder, controlling and steering devices, sources of plasma gas and gas supplying the feeder, sampling system.

In the experiment an arc plasmatron was used. Plasma jet was generated by argon and mixture of Ar and H₂ (maximum 3.6 %). Hydrogen was used to stabilize the arc and to stimulate amount of produced hydrocarbons.

![Diagram of research test-stand](image)

For the purposes of the experiment rubber waste from retreading facility was used. The rubber was powdered to 250 μm particles. Only rubber material of this size particles guarantees its full decomposition in the plasma jet [2].

3. Experimental

The experimental research of plasma pyrolysis of rubber has been carried out in Ar plasma jet and Ar + 3.6 % H₂ plasma jet. The amount of H₂ was limited to the level of 3.6 % due to safety measures.

During pyrolysis in the Ar plasma jet the mass flow rate of rubber powder and flow rate of Ar were constant. They were equal: 2.29 kg/h and 5325 l/h, respectively. The electric power of the plasmatron was changed in the range from 9.6 kW to 20.35 kW. The change of the plasmatron power was obtained...
by change of the electric current in the circuit of the DC power supply unit.

For Ar + 3.6 % H2 plasma gas the mass flow rate of the rubber powder was constant and equalled 2.29 kg/h. The flow rate of the plasma gas was also constant and it was equal to 5453 l/h. The electric power of the plasmatron changed in the range from 11.7 kW to 26.95 kW. The bigger power of the plasmatron for the Ar + 3.6 % H2 plasma was caused by the presence of H2 in the plasma gas. It is caused by molecular character of hydrogen.

In the case that each test gas products were collected triply to ensure repeatability of the research. The samples were collected in glass test-tubes. Prior to the tests the test-tubes has been vacuumed.

The samples of the gas products were analyzed by the IR absorption spectroscopy method [2].

4. Results

In the absorption spectra of the collected samples of the Ar plasma pyrolysis bands of the following gases were identified: CH4, C2H2, CO, CO2. The gaseous products must have contained also H2: [1], but the applied method made it impossible to detect.

The increase of the plasmatron power caused rising production of all the identified gases. The courses of concentration were monotonic. Exemplary, Fig. 2 presents the course of production of C2H2.

Figure 2: Concentration of C2H2 in gaseous products for Ar plasma in function of plasmatron power (Q=5325 l/h, g=2.29 kg/h, I=300 ÷ 550 A)

In spectroscopy spectra there was not detected presence neither of NOx nor SO2 compounds. It is important and promising because NOx are inherent products of all the utilization methods based on conventional incineration and thermal pyrolysis.

The analysis of the absorption spectra of the products of the Ar + 3.6 % H2 pyrolysis revealed the presence of CH4, C2H2, CO, CO2, likewise for the Ar plasma pyrolysis. However, there were also detected bands of two additional gases. For the purpose of the research they have been called Alfa 1 and Alfa 2. With a high probability, approaching certainty, they are propane and butane.

The presence of H2 in the plasma gas clearly caused the increase of the plasmatron power.

Similarly, like for Ar plasma pyrolysis, there was found no band of NOx and SO2 in the gaseous products.

The increase of the plasmatron power caused the monotonic increase of all the gaseous products concentration. The exemplary course of concentration of C2H2 for the Ar + 3.6 % H2 plasma pyrolysis products presents Fig. 3.

A small amount of soot was also the product of the pyrolysis of the rubber waste for Ar plasma as well as Ar + 3.6 % H2 plasma. The amount of the soot did not exceed several thousandth of percent.

5. Conclusions

The experiments on the plasma pyrolysis of rubber waste helped to form the following conclusions:
- all delivered rubber powder was entirely decomposed,
- products consisted of mainly gas and small amount of soot,
- differences occurred between Ar and H2 plasma pyrolysis,
- H2 in plasma gas caused production of larger number of hydrocarbons compared to clean Ar plasma,
- H2 in the plasma gas increased plasmatron power,
- products concentration for Ar + 3.6 % H2 plasma is bigger than for Ar plasma,
- no presence of toxic compounds, like dioxins, NOx, nor SO2 was identified in the plasma pyrolysis products. A very high process temperature made it possible thermal treatment of the rubber and yet no harmful products were identified which had been inevitable in case incineration,
- small amounts of sulfur and metals were contained in the soot and they did not leave the reactor,
- increasing electrical power of the plasmatron caused monotonically rising concentration of all the products of the plasma pyrolysis, for both types of plasma gas.

Considering the results of the plasma pyrolysis of rubber waste there comes the main conclusion that the described method is the effective one and environment friendly. An economic aspect of the process can be improved by energy generation out of gaseous products of the plasma pyrolysis of rubber.

References

Numerical analysis of thermal decomposition of a single solid fuel particle in a stream of hot flue gases

Izabela Wardach-Święcicka¹, Dariusz Kardaš²

¹, ²Renewable Energy Department, Institute of Fluid-Flow Machinery Polish Academy of Sciences
Fiszera 14, 80-231 Gdańsk, Poland
e-mail: izkaw@imp.gda.pl, dk@imp.gda.pl

Abstract

The aim of the work is to investigate the heat and mass transfer during pyrolysis of a single solid fuel particle. It regards the thermal decomposition process which occurs in the absence of oxygen. Basing on the mathematical modeling and experimental data the in-house code for the devolatilization process was obtained. The model consists of mass, momentum and energy equation for solid and gas phases. Numerical simulation results show that for various fuel physical properties (volatile content, moisture content, particle size) the different times of particle devolatilization were obtained. Moreover, a total heating time strongly depends on numerical model assumptions too. The numerical results were compared with the experimental data available in the literature, showing a good qualitative agreement.

Keywords: numerical analysis, heat and mass transfer, fuel single particle, devolatilization

1. Introduction

Combustion of solid fuel particles plays an important role in energy generation. In Poland more than 90% of electrical energy is produced from coal. The mass transfer during heating of the solid fuel is the basic phenomenon in the first stage not only of the combustion but also of gasification processes, in which the alternative fuels like gas, liquids or solids can be produced. Coal or biomass may have different physical and chemical properties, depending on mining/growing place. This affects the efficiency of energy or fuel production. Moreover, the composition of final product may vary and thus operating of the reactor, coke oven or boiler becomes even more difficult. Therefore, an analysis of devolatilization of single solid fuel particle has a practical meaning. Moreover, the behaviour of solid particles with a mass loss phenomenon is interesting from research reasons. Due to the heat transfer from hot flue gases to a fuel particle, mass exchange from particle to its neighbourhood takes place. At the beginning of the heating process water vaporizes from the particle surface. In the next phase of heating, pyrolysis causes changes of the internal structure; in consequence, volatiles are released as gases. In the combustion at its final stage, carbon included in the particle undergoes surface reactions with gases. These processes lead to mass, volume and shape changes [1], which have an important influence on the residence time of particles in the reactor chamber. These parameters affect the particle conversion/burnout degree, which determines the whole process efficiency.

Numerical modeling is practically the only way of broadening the knowledge on devolatilization of combustion of fine solid particles. It gives detailed information about temperature and velocity fields inside and in the vicinity of particles. In the work devolatilization process was taken into account only.

2. Devolatilization modeling

Numerical simulations concern pyrolysis a process of a single solid particle which occurs as a consequence of the particle temperature increase. The aim of the research is to investigate the impact of particle physical properties on the devolatilization process. In the mathematical modeling of pyrolysis process the fuel grain is considered an ideal sphere consisting of a porous solid phase and a gaseous phase. An additional assumption is made on physical properties to be functions of the radial variable, simplify the final form of the partial differential equations. The model consists of the mass, momentum and energy equations for a porous spherical solid particle heated by a hot gas stream, as follows:

– mass balance equation for solid and gas phase:

$$\frac{\partial \varepsilon_s \rho_s}{\partial t} = -\mathbf{S}_s, \quad \frac{\partial \varepsilon_g \rho_g}{\partial t} + \frac{1}{r^2} \frac{\partial}{\partial r} \left(r^2 (\varepsilon_g \rho_g u)\right) = \dot{S}_g,$$

where \(\varepsilon_s \ [m^3/m^3]\) is the volume fraction of solid phase, \(\rho_s \ [kg/m^3]\) is the density of solid phase, \(S_s \ [kg/m^3/s]\) is the mass source for solid phase, \(\varepsilon_g \ [m^3/m^3]\) is the volume fraction of gaseous phase, \(\rho_g \ [kg/m^3]\) is the density of gaseous phase, \(S_g \ [kg/m^3/s]\) is the mass source for gaseous phase and \(r \ [m]\) is the radial position,

– momentum equation for pyrolysis gases:

$$\frac{\partial \varepsilon_g \rho_g u}{\partial t} + \frac{1}{r^2} \frac{\partial}{\partial r} \left(r^2 (\varepsilon_g \rho_g u \otimes u)\right) = -\frac{\partial \varepsilon_g \rho_g}{\partial r} - \mu \varepsilon_g \frac{\partial T}{\partial r},$$

where symbol \(u \ [m/s]\) denotes gas velocity, \(p \ [Pa]\) is the gas pressure, \(\mu \ [Pa \cdot s]\) is the dynamic viscosity and \(K \ [m^2]\) is the permeability,

– energy balance with the assumed local thermal equilibrium state between the phases:

$$\frac{\partial \rho_{eff} \varepsilon_{eff} T}{\partial t} + \frac{1}{r^2} \frac{\partial}{\partial r} \left(r^2 (\varepsilon_{eff} \rho_{eff} T u)\right) = -\varepsilon_g \frac{1}{r^2} \frac{\partial}{\partial r} \left(u^2 u\right) - \frac{1}{r^2} \frac{\partial}{\partial r} \left(2 \lambda_{eff} \frac{\partial T}{\partial r}\right) + \Psi,$$

where \(T \ [K]\) is the temperature of gas and solid phase, \(\lambda \ [W/mK]\) is the thermal conductivity, \(\varepsilon \ [kJ/kgK]\) is the specific heat, \(\Psi \ [kJ/m^3/s]\) represents the internal volumetric heat sources, subscript \(eff\) denotes the effective values. For the closure of the equation system presented above, the additional ideal gas state equation is taken into account.
Comparing to the model described in [2], the presented model accounts also for the vaporization of the fuel [3]. The mass source term for solid and gas phase is determined in the wide range of temperature according to experimental data (IChPW Zabrze [4]). The devolatilization rate is defined by the Arrhenius formula [5]. The proposed gas release model is limited by the state of devolatilization process reached at slow heating rates [6], which allows to determine the pyrolysis process direction. In the work two approaches of energy balance equation were studied: with and without convection of gases.

3. Results

The results of numerical analysis of a pyrolysis process of dry coal are presented in Table 1. Two cases were analyzed: with and without the convective term in the energy balance equation (3). It may be observed, that different fuel physical properties (particle size) lead to various times of conversion.

### Table 1: Numerical results of conversion time for different coal particles. The velocity and temperature of surrounding gas: \( u_{fluid} = 10^{-4} \text{m/s}, T_{fluid} = 1273 \text{ K} \). Initial temperature of fuel \( T_0 = 300 \text{ K} \), initial porosity \( \epsilon_g = 0.236 \).

<table>
<thead>
<tr>
<th>diameter ( d_p ), m</th>
<th>90% conversion</th>
<th>no convection</th>
<th>convection</th>
<th>no convection</th>
<th>heating</th>
<th>convection</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.001</td>
<td>6.03</td>
<td>8.34</td>
<td>42.15</td>
<td>45.35</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.002</td>
<td>20.01</td>
<td>26.46</td>
<td>139.74</td>
<td>152.03</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.003</td>
<td>37.57</td>
<td>49.73</td>
<td>258.24</td>
<td>281.27</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure 1 presents a temperature profile in a dry coal particle \( d_p = 0.0102 \text{ m}, \epsilon_g = 0.0236 \), comparison with experiment.

![Temperature profile in a dry coal particle](image1)

Figure 1: Temperature profile in a dry coal particle \( d_p = 0.0102 \text{ m}, \epsilon_g = 0.0236 \), comparison with experiment.

A total heating and conversion time strongly depends not only on the physical properties of fuel, but on the numerical model assumptions too. Generally devolatilization and heating time depend on physical and chemical properties of the fuel (volatile and moisture content, particle size) and on the pyrolysis process conditions.

### References


Changing the combustion area of powder grains in the engine of a two-chamber system

Zbigniew Wrzesiński
Faculty of Production Engineering, Warsaw University of Technology
ul. Narbutta 85, 02-524 Warszawa, Poland
e-mail: zw@imik.wip.pw.edu.pl

Abstract

The discussion presented in this article concerns a part of the question connected with the so-called main problem of interior ballistics (MPIB) of two-chamber systems. The solution of the main problem of interior ballistics (MPIB) of two-chamber systems consists mostly in defining, in the function of time, the development of pressure, temperature of a gas/powder mixture (GPM) and gaseous combustion products (GCP) in the generator and engine of a two-chamber system. The physical model of the main problem of interior ballistics (MPIB) of two-chamber systems as well as its mathematical description are presented in [1] and [2]. This part of the physical model and its mathematical description which relates to the flow of a gas/powder mixture (GPM) into the engine when the membrane in the generator is ripped is the subject of the present discussion. It is quite a different, alternative approach when compared to the existing ones. It has been adopted as a result of encouragement provided by rapid development of modern computation technology equipment creating favourable conditions for building more advanced physical models and their mathematical descriptions which may now be effectively presented.

Keywords: combustion area of powder grains, two-chamber system, the main problem of interior ballistics

1. Introduction/problem

In a two-chamber system, when powder grains during a shot can move from the generator to the engine, a problem arises how to define the changing combustion area of powder grains which have flown from the generator to the engine.

2. Balance of the Extensive Values

Because the combustion area of powder grains in the engine of a two-chamber system is an extensive value (EV), we can apply the balance axiom which says:

The EV in a balance system can change only as a result of the production (P) of EV in a balance system or the change (CH) of EV through the border of a balance system or a result of both processes undergoing at the same time.

Taking into account the fact that EV production in a balance system is equal to the sum of the creation (C) of EV, which is always negative or equal to zero, we can write as follows:

\[ C \geq 0 + A \leq 0 = P \leq 0 \]  
\[ \text{EVcreation} \quad \text{EVannihilation} \quad \text{EVproduction} \] (1)

It stems from the above that EV production can be positive or negative depending on which component of equation (1) prevails.

The change of EV through the borders of a balance system is equal to the sum of the creation (C) of EV, which is always positive or equal to zero, and the outflow (OF) of EV from the system, which is always negative or equal to zero. Thus we can write:

\[ I \geq 0 + OF \leq 0 = CH \leq 0 \]  
\[ \text{EVinflow} \quad \text{EVoutflow} \quad \text{EVchange} \] (2)

The exchange may take on different positive or negative values depending on which of the Eqn (2) elements prevails.

Taking into account the balance axiom for extensive values (EV), we will write the basic balance as follows:

\[ \{ EV \}_{\text{change}} = \{ EV \}_{\text{production}} + \{ EV \}_{\text{exchange}} \] (3)

The segment of production P of the powder grain combustion area in the balance system of a two-chamber system engine is equal only to annihilation A as the creation C of the powder grain area in the engine

\[ C = 0 \] (4)

Hence the production segment P of the powder grain combustion area equals annihilation A of the powder grain combustion in the engine

\[ P = A \] (5)

The exchange segment E of the powder grain combustion area in an engine balance system of a two-chamber system is only equal to the inflow IF because the outflow segment OF of the engine is

\[ OF = 0 \] (6)

Hence the exchange segment E of the powder grain combustion is equal to the inflow IF of the powder grain combustion area to the engine

\[ E = IF \] (7)

As it stems from the above reasoning, the change in the powder grain combustion area in the engine of a two-chamber system S equals its annihilation A in the engine plus its inflow IF to the engine from the generator, which is described by the balance equation below.

\[ S = A + IF \] (8)

The above balance equation makes it possible to proceed to an analysis of quantitative changes in the powder grain combustion area in the engine of a two-chamber system.

The diagram of a two-chamber system presented in Fig.1 consists of the following elements:
1 – a high pressure chamber called generator
2 – a low pressure chamber call engine
3 – inflow holes bored in the generator
4 – a membrane covering the holes, placed inside the generator
5 – a mobile piston
6 – an ignition mechanism placed inside the generator
7 – powder grains
8 – a primer
9 – mercury fulminate
10 – a firing pin

Figure 1: Two-chamber system

The following physical model of interior ballistics was formulated for a gas/powder mixture (GPM) in a two-chamber system.

The volume of a gas/powder mixture (GPM) in the generator at the time that the membrane bursts is constant.

\[ V = V_0 \Rightarrow \Delta V = 0 \quad (9) \]

Where:
- \( V \) – volume of a GPM mass
- \( V_0 \) – initial volume of powder mass
- \( p_{g} \) – pressure of gaseous combustion products in the generator
- \( p_{g_{m}} \) – pressure of gaseous combustion products at which the membrane in the generator bursts

The moment that the membrane bursts there is an outflow from the generator to the engine. The volume of GPM mass changes then within the limits

\[ V_0 - Y \leq m \leq V_0 \quad (10) \]

where \( Y \) is the volume of GPM mass which had flown from the generator to the engine until the powder in the generator stopped.

Assuming the following notation:
- \( N_P \) – initial quantity of powder grains of the propellant charge in the generator (before the membrane bursts)
- \( N \) – current quantity of powder grains of the propellant charge in the generator (after the membrane bursts)
- \( N_g \) – quantity of powder grains of the propellant charge in the generator when the powder stops burning in the generator
- \( n \) – quantity of powder grains of the propellant charge which has flown from the generator to the engine

A quantitative balance was formulated for powder grains in the propellant charge in the generator in a generator-engine system.

For the sake of the described physical model of interior ballistics GPM in a two-chamber system the following two theorems were formulated.

The first of them is that of distribution. It says that when a shot is fired the area of quantitative density of powder grains in the generator is homogenous (gradientless) and non-stationary.

The second theorem is that of relative volumes. It postulates that when a shot is fired relative volumes of

\[ \frac{\omega_{p} - Y}{\omega_{p}} \quad (12) \]

powder grains

\[ \frac{N}{N_{p}} \quad (13) \]

in the generator are equal, where \( Y \) – signifies the current volume of GPM mass flown from the generator to the engine.

Taking into account the distribution theorem and that of relative volumes, we can express the following relationship:

\[ \frac{N}{N_{p}} = \frac{\omega_{p} - Y}{\omega_{p}} = 1 - \frac{Y}{\omega_{p}} \quad (14) \]

Because it has been assumed that the pressure area of gaseous combustion products in the generator is homogenous and non-stationary, the dimensions of all powder grains in the generator will change in time in the same way, burning according to the geometrical law of combustion.

However, the grains flowing out of the generator and into the engine after the membrane bursts will vary in size because their combustion times in the generator will be different. The grains which burnt shorter in the generator will be geometrically larger than those that burnt in the generator longer.

References


Laboratory investigation of the influence of pipeline supports stiffness on water hammer and fluid-structure interaction

Adam Adamkowskii1, Slawomir Henclik1, Waldemar Janicki2, Mariusz Lewandowski4*

1,2,3*THe Szewalski Institute of Fluid-Flow Machinery, Polish Academy of Sciences, Fiszera 14, 80-231 Gdansk, Poland

e-mail: aadam@imp.gda.pl 1, shen@imp.gda.pl 2, wjanicki@imp.gda.pl 3, mlew@imp.gda.pl 4

Abstract

Water hammer (WH) is produced when steady pipe flow conditions are disturbed by any reason, e.g. valve operation, hydraulic machinery load variation, etc. In the classic theory of WH the pipe is assumed rigid and the fluid-structure interaction (FSI) is considered a quasi-static effect. This assumption is a good approximation in many real cases. However, when the pipe is flexible or fixed to the foundation with elastic supports then the pipe motion forced by the unsteady flow influences in reverse the flow and the dynamic FSI should be taken into account. A special test rig for WH-FSI experiments was designed and built in the laboratory hall of the Institute of Fluid-Flow Machinery PAS. The laboratory pipeline was fixed to the foundation with a number of elastic supports and a WH was produced by a sudden valve closure. The pressure changes, pipe stresses and pipe motion during the transient were measured for various flow initial conditions. These measurements were repeated for various types of supports and the influence of the support stiffness on pressure changes and pipe stresses were analyzed and concluded.

Keywords: closed conduits, pipelines, hydraulic transient, liquid unsteady flow, water hammer, fluid-structure interaction

1. Introduction

Water hammer is the result of a sudden change in steady pipe flow conditions due to valve operations or other reasons. In the classic theory of WH [1, 7] the interaction between the fluid and the structure in a closed conduit is considered quasi-static, i.e. an elastic deformability of a pipeline coating due to internal pressure is taken into account within the expression for the celerity of pressure wave. In such an approach three main effects [4, 6] connected with dynamic fluid-structure interaction are neglected: (1) the Poisson coupling effect, (2) the friction coupling effect caused by the relative motion of the liquid and the pipe wall, and (3) the influence of bends, conduit supports and the flow throttling elements (e.g. constrictions) in the pipeline, which produce the junction coupling effect.

In the available WH-FSI literature [5], especially within the field of experimental verification [2, 3] there is no sufficient recognition of the influence of FSI effect on WH. It is usually concluded that for rigidly fixed conduits and sudden changes in the flow the classical approach of water hammer gives sufficiently reliable results of predicted changes in the fluid pressure and stress in the material of the pipeline coatings. On the other hand, in the case of flexible pipelines, particularly for stepwise flow changes, numerical predictions based on the classical approach are excessively inaccurate due to neglecting FSI effects.

2. Laboratory pipeline

In the laboratory hall of the Institute of Fluid-Flow Machinery (IMP PAN) in Gdansk a special laboratory rig was designed and built. The main part of the rig was a 60 m long copper pipeline built of several straight pipe reaches joint by knees and fixed to the foundation with elastic supports. The pipeline was dedicated for testing the influence of the support stiffness on pressure changes and pipe stresses during a simple WH event. The pipeline scheme is presented in Figure 1.

![Figure 1: The scheme of the laboratory pipeline.](image1)

The tests were performed for different stiffnesses and configuration of pipeline supports for different flow initial conditions. The supports were specially designed and manufactured at IMP PAN and their stiffness matrices were calculated. The results for three types (FS2, FS3, FS4) of supports shown in figure 2 are presented in the paper. The relative stiffness of the supports is approximately 1:4:16.

![Figure 2: The pipe supported with (from left) FS2, FS3, FS4.](image2)

Steady flow was driven by a constant pressure of the pressure vessel at the beginning and the opening rate of the control valves. The WH was excited by a sudden closure of the...
shut-off valve at the end. The phenomenon was observed by measuring the pressures, pipe stresses and pipe motion during decaying of the transient. The four pressure transducers were situated at about every quarter of the pipe length (starting from the valve back). Some other auxiliary quantities (pressure at the vessel, initial flow velocity, temperature, valve closing rate) were also registered.

3. Measurement results and conclusions

The measurements were conducted for ten different initial flow velocities and for two static pressures of the pressure vessel. Different types of pipe supports were used in two types of their arrangement. The valve at the end of the pipe was closed rapidly with the use of special spring drive within about 5 msec. A large amount of data was registered and preliminary analyzed. A further detailed analysis is still intended.

In figures 3 and 4 selected results for FS2, FS3, FS4 supports mounted in same arrangement (configuration denoted as L) and rapidly closed valve are presented. In figure 3 due to the initial conditions marked as W7 (initial flow velocity v=0.7m/s, initial pressure in the upper reservoir p=1.12 MPa) the WH takes place without a liquid column separation.

In figure 4 the initial conditions marked as N9 (v=1.4m/s, p=0.72 MPa) involve the WH with column separation.

![Figure 3: Pressure run at the valve for initial conditions W7 (no liquid column separation).](image)

![Figure 4: Pressure run at the valve for initial conditions N9 (liquid column separation).](image)

4. Summary

Experimental testing of water hammer and a fluid-structure interaction at a special laboratory pipeline built at IFFM PAS are described in the paper. Selected measurement results are presented and preliminary conclusions are formulated. At the first approach all these effects can be understood by a proper physical analysis. However further detail investigations are intended in order to match the observed effects with the theoretical models.

References


Technical coefficients for continuum models of orthotropic tensegrity modules

Anna Al Sabouni-Zawadzka¹, Wojciech Gilewski²

¹,² Faculty of Civil Engineering, Warsaw University of Technology
Al. Armii Ludowej 16, 00-637 Warsaw, Poland
e-mail: sabouni@il.pw.edu.pl¹, w.gilewski@il.pw.edu.pl²

Abstract

The paper focuses on application of continuum models for orthotropic tensegrity modules and the determination of their technical coefficients. Tensegrities are cable-strut systems with a special node configuration, which ensures the occurrence of infinitesimal mechanisms balanced with self-stress states. The continuum model of the module is built by assuming that the strain energy of the unsupported tensegrity structure is equivalent to the strain energy of the cube. After the proper validation, the proposed model can be used to determine and interpret physical properties of the module. From the obtained elasticity matrix and the inverse matrix, the technical coefficients such as: Young’s moduli, Poisson’s ratios and shear moduli can be computed.

Keywords: tensegrity structures, linear elasticity theory, 3D continuum model, technical coefficients

1. Introduction

The term “tensegrity” was first introduced by Buckminster Fuller (see [5] for historical details). Several definitions of this concept can be found in the literature [4]. For the purpose of this paper, a tensegrity structure is defined as a pin-jointed system with a particular configuration of cables and struts that form a statically indeterminate structure in a stable equilibrium. Tensegrities consist of a discontinuous set of tensioned elements inside a continuous set of tensioned members, the latter of a zero compressive stiffness. Infinitesimal mechanisms, that occur in tensegrity structures, are balanced with self-stress states. The major advantages of tensegrity systems are: large stiffness-to-mass ratio, deployability, reliability and controllability [4,5].

The continuum model of tensegrity modules should allow to:
- estimate properties of the module with typical deformation modes (tension, shear),
- evaluate the influence of self-stress for a defined deformation,
- evaluate the influence of cables and struts for the properties of the module,
- compare elastic properties of typical tensegrity modules,
- determine the technical coefficients,
- find a physical interpretation for the technical coefficients.

2. 3D continuum model

Discrete models of tensegrity modules are mathematically described with the use of the Finite Element Method [6]. The strain energy is a quadratic form of nodal displacements q:

\[ E_{s}^{FEM} = \frac{1}{2} q^T Kq \]  \hfill (1)

with the global linear and geometric stiffness matrix \( K=K_{s}+K_{G} \) as a kernel. The self-stress of the module (proportional to the tension force \( S \)) is represented by the geometric stiffness matrix. In order to build a continuum model of tensegrity modules, the symmetric linear 3D elasticity theory is considered. According to this theory, the strain energy can be expressed as:

\[ E_{s}^{LES} = \frac{1}{2} \varepsilon^T E \varepsilon \ dV \]  \hfill (2)

where: \( \varepsilon \) – the strain vector, \( E \) – the elasticity matrix.

To compare the energies and build the equivalent matrix \( E \), the nodal displacements are expressed by the average mid-values of displacements and their derivatives with the use of Taylor series expansion. Coordinates of nodal points \{\( x_{i} a \), \( a \)\} are expanded in Taylor series by the edge length \( a \) with the increments: \( \Delta x = a, \Delta y = a, \Delta z = a \).

The obtained elasticity tensor can be expressed in Voight’s form \( E_{ij} = [e_{ij}]_{i,j=1,2,...,6} \). There are 21 independent coefficients for anisotropy. However, the analyses presented in this paper have been limited to orthotropic cases with nine independent coefficients.

3. Model validation

A proposed continuum model was validated by comparing several results obtained from this model with the corresponding results obtained from the truss analysis. The validation was performed on an orthotropic space truss. The truss was tested for the technical parameters: Young’s modulus \( E \), Poisson’s ratio \( \nu \) and shear modulus \( G \). Nine independent coefficients were determined and compared. Figure 2 shows the test which was carried out in order to determine the Young’s modulus and the Poisson’s ratios for one direction. The same tests were performed for the other two directions.

Figure 1: Tensegrity and continuum
The above conditions lead to the limitations of the obtained technical coefficients: a) Young’s moduli, Poisson’s ratios and shear moduli were computed. Shear moduli depend linearly on the parameters.

In case of the positive definite matrices, their principal minors are positive. This imposes limitations on the obtained technical coefficients: $E_{11}, E_{22}, E_{33} > 0$, $G_{12}, G_{13}, G_{23} > 0$, $v_{12}v_{13} + v_{23}v_{13} + v_{32}v_{13} < 1, v_{23}v_{13} < 1, v_{13}v_{13} < 1$. The above conditions lead to the limitations of the self-stress, which is expressed by the value $\sigma$.

In order to determine technical coefficients of the module, several operations were performed. Shear moduli can be computed directly from the components of the elasticity matrix $E$: $E_i = 1/e_{ii}$, $E_{ij} = 1/e_{ij}$, $v_{ij} = -e_{ij}e_{ii}/E_{ii}$, $v_{ij} = -e_{ij}e_{ii}/E_{ii}$, $v_{ij} = -e_{ij}e_{ii}/E_{ii}$, $v_{ij} = -e_{ij}e_{ii}/E_{ii}$.

5. Conclusions

Tensegrity structures are complicated regarding both their geometry and mechanics. In order to understand their properties and identify technical coefficients, a continuum model is suggested. The continuum model of the orthotropic tensegrity module – an expanded octahedron – was built assuming that the strain energy of the unsupported tensegrity structure is equivalent to the strain energy of the cube. After the proper validation, the proposed model was used to determine and interpret physical properties of the module. From the obtained elasticity matrix and the inverse matrix, the technical coefficients such as: Young’s moduli, Poisson’s ratios and shear moduli were computed.

Similar analysis will be performed for other types of symmetry and for anisotropic systems.

References

Experimental tests for the determination of mechanical properties of PVC foil

Andrzej Ambroziak
Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: ambrozan@pg.gda.pl

Abstract

The paper presents an application of the PVC film in civil engineering. Mechanical behaviour of the PVC film applied for suspended ceilings, in the form of a stretch ceiling is investigated under uniaxial and biaxial tensile tests. The study is focused on the determination of mechanical properties from experimental data (uniaxial and biaxial tensile tests). The uniaxial cyclic tests are performed in order to observe a variation of immediate mechanical properties under cyclic load. This paper is suggested to be an introduction to a comprehensive investigation on civil engineering application of the PVC foils.

Keywords: PVC film, stretch ceiling, mechanical properties, mechanical test, uniaxial tests, biaxial tests

1. Introduction

The applications of plastics are widespread in various branches of industry. Sorts of polyvinyl chloride (PVC) films are used in stretch ceilings. The application of flexible PVC films in stretched ceilings is a modern form of design and interior decoration. Stretched film ceiling installation consists of a lightweight, highly durable film or a decorative fabric to a framing construction (aluminium or steel-aluminium) intended to hold it stretched in place. The architectural applications of flexible PVC films are still being developed.

Recent developments in membrane building materials for lightweight structures were described by Saxe [1]. The advantages of lightweight tensioned coated fabrics and foils applied to the existing building sector in order to improve the insulating/shading performance of the external building envelopes were reviewed by Baccarelli and Chilton [3]. Another types of foils made of ETFE is used in civil engineering for tensile structures e.g. roofs and claddings. Mechanical behaviour of ETFE foils under uniaxial and biaxial loading was investigated by Galliot and Luchsinger [2].

The building site supervision makes it possible to recognize new technologies and applications of new materials. The author performs construction site supervision on the Alchemia building site in Gdansk where the PVC foil is applied for stretched ceilings. The Alchemia building is a modern multi-purpose complex in Gdansk, Poland. It covers office spaces and supplies a sports and recreation facility offering a 25-meter swimming pool, a sports hall for team sports, fun climbing zone, a gym and other facilities and attractions. The Alchemia swimming pool is covered by a stretched ceiling. The PVC foil is suspended on a steel-aluminium structure.

2. Laboratory tests

Uniaxial and biaxial tensile tests are chosen from a large group of experimental tests (see e.g. [4]) to model the material behaviour of a PVC foil. The aim of the laboratory tests is to determine mechanical properties of a polymer, named DPS stretch ceiling. According to the technical data specified by the manufacturer (http://www.grupadps.com/en/) the following properties are defined: weight 240g/m², total thickness 0.17mm, tensile strength for longitudinal and transverse directions 17.1N/mm² and 18.7N/mm², respectively. The tensile strength is established according to PN-EN ISO 527-1:1998. Material parameters necessary to analyze a real structure were not found in technical specification. In order to perform a relevant assessment of a material, laboratory tests were carried out.
For a detailed comparison of the uniaxial cyclic tests results, the residual strains $\varepsilon_{\sigma=1}, \varepsilon_{\sigma=10}, \varepsilon_{\sigma=20}$ (where $n$ is a number of the cycle) and longitudinal stiffness values $F_n=1, F_n=10, F_n=20$ were assigned. It should be noted that the longitudinal stiffness parameter $F$ (called tension stiffness) is specified in [N/m], in the case of uniaxial tensile tests corresponding to Young’s modulus. The unit [N/m] is accepted in accordance with the units generally used for fabrics.

The results of the identification are collected in Table 2. The values of $F$ [N/m] parameters are specified for stress ranges $>100\text{N/m} (>5\%$ of the ultimate tensile strength - UTS). Observation for each stress level yields that the increment of cycle number grows makes the longitudinal stiffness increase ($F_n=20 > F_n=10 > F_n=10 > F_n=0$). The immediate mechanical parameters ($F_n=0$) change during the cyclic tests in the analysed range of loads. The values of residual strains $\varepsilon_{\sigma=20}$ range between 3% and 20%.

### Table 2: Results of identification – cyclic tests

<table>
<thead>
<tr>
<th>Tests type</th>
<th>Stress values [N/m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>200</td>
</tr>
<tr>
<td>B</td>
<td>400</td>
</tr>
<tr>
<td>C</td>
<td>800</td>
</tr>
</tbody>
</table>

It should be noted that the immediate properties ($n=0$) specified in Table 2 refer to the initial stress state during the erection process. These parameters are indispensable in the film cutting pattern process too. The concern is, how the PVC film behaves in each direction at the initial stress state. Additionally, the behaviour of PVC film under cyclic loads should be specified. These tests show the variation of immediate mechanical properties under cyclic loads.

### 2.2. Biaxial tensile tests

The biaxial tensile tests performed on the Zwick system with a video extensometer attached (see Figure 2), were made for the cross-shaped specimens. The arm width was 100 mm, therefore on the testing area of 100x100 mm the gage length of about 50 mm in both directions was indicated. Due to all the tests initial grip separation of 300 mm was selected. The specimens were subjected to cyclic tension (constant force rate) in longitudinal and transverse directions with load (stress) ratios 1:1, 1:2, 2:1 ($\sigma_{\varepsilon_1} : \sigma_{\varepsilon_2}$, where $\sigma_{\varepsilon_1}$ and $\sigma_{\varepsilon_2}$ are stresses in the longitudinal and transverse directions, respectively).

For the sake of engineering calculations it is possible to assume the isotropic film theory. This theory yields the following:

$$\sigma_{\varepsilon_1} = \frac{F}{1-\nu} (\varepsilon_{\varepsilon_1} + \nu \cdot \varepsilon_{\varepsilon_1}), \quad \sigma_{\varepsilon_2} = \frac{F}{1-\nu} (\varepsilon_{\varepsilon_2} + \nu \cdot \varepsilon_{\varepsilon_2})$$

where $\nu$ is the Poisson’s ratio, $F = (F_1 + F_2)/2$, $\varepsilon_{\varepsilon_1}$, $\varepsilon_{\varepsilon_2}$ are strains in transverse and longitudinal directions, respectively. An isotropic model is frequently used for the analysis of suspension structures made of plastic film, rubber-like materials, etc.

Assumed $F=7280\text{N/m}$ (the mean value of $F_n=0$ for A range, see Table 2) the value $\nu=0.4$ according to Equation 1 is specified.

### 3. Conclusions

The study presents test methods to investigate the mechanical properties of PVC foils. A modern laboratory equipment allows for different variants of tests and the computer storage of the results, important for a future identification process. The comparison of test results on the same material from different laboratories detects their discrepancies. The procedure of manufacturing PVC films may results in variation of mechanical properties. Examples of such tests were presented in the paper in order to understand the PVC film behaviour better.

The research program completed by the author for the construction of new stretched ceilings of the Alchemia building in Gdańsk presents various applications modern testing devices. Such tests are necessary in all stages - before, during and after construction. The tests results may be used in the solution of problems inherent to the roof assembly and its service. The investigation confirms that the quality of the PVC foil, equipment and systems is sufficiently high.

### References


On a four-time unification of Cosserat continua by the intrinsic approach

Janusz Badur¹, Jacek Chróścielewski²

¹Department Conversion Energy, Institute of Fluid – Flow Machinery PAS
Fiszera 14, 80-231 Gdańsk, Poland
e-mail: janusz.badur@imp.gda.pl

²Chair of Structures Mechanics, Technical University of Gdańsk
e-mail: jchrost@pg.gda.pl

Abstract

Two original Cosserat’s concepts: the four-time unification and the intrinsic formulation based on the internal von Helmholtz group are revalorized and explained in terms of the modern mechanics.

Keywords: Cosserat continua, Cosserat D–brane, four–time formulation, von Helmholtz intrinsic symmetry.

1. Introduction

The intrinsic formulation of continuum mechanics comes from von Helmholtz old concept concerning an additional local space-time symmetry. This symmetry, frequently called the intrinsic symmetry, usually leads to additional degrees of freedom and additional internal nonlinearity. The concept of local symmetry itself is a way of introducing into description hidden parameters that cannot appear within the framework of global symmetry and the classical Lagrange or Euler description of continua.

This new possibility mainly comes from the fact that the intrinsic formulation needs to postulate some local symmetry described by a continuous Lie group. The Lie algebra generated by this intrinsic symmetry should be a base where an observer is located now. This observer cannot measure the classical elements known from Euclidean geometry, therefore a new type of continuum geometry would be developed. Such a continuum geometry, compatible with the space-time arena was firstly located now. This observer cannot measure the classical elements known from Euclidean geometry, therefore a new type of continuum geometry would be developed. Such a continuum geometry, compatible with the space-time arena was firstly developed by Darboux and next by Elie Cartan. However, only in Cosserat’s monograph [3], von Helmholtz concept of intrinsic group of symmetry has been finally stated and completely so applied to continua of different dimensions. In a short time period of 1909-1936, owing to effort of such scientists like Poincaré, Appel, Roy, H. Cartan, Sundria, the method of intrinsic formulation diffused into the field theory, especially to electrodynamics and gravitation.

Unfortunately, each of these developments ran independently, losing a main Cosserat idea concerning a four-time unification. Therefore, in this report is undertaken a problem of revalorization of the common description of zero-, one-, two- and three-dimensional continua.

2. Cosserat’s intrinsic formulation

This method is based on a simple extension of the natural referential, curvilinear coordinate system \(x^M, M = 1,2,3\) having a time \(t\), to a common system called “the four-time”. Then the referential base system should be defined as: \(G^i, G^{αβ}\) for three-dimensional body. \(G^i, G^{αβ}\) for three-dimensional body. Let a referential gradient operator be defined:

\[
\text{Grad}_i = (\partial_i \otimes G^{αβ} + \partial_{αβ} G^i + \partial_i G^{αβ}) \text{Grad}_α \text{Grad}_β.
\]

(1)

The intrinsic placement is usually described in terms of Lie algebra base adequate to the intrinsic group. Let us, for the von Helmholtz group take for simplicity the local Cosserat reper base \(d_α, d_β, d_γ = d_a, d_a, α = 1,2,3\). In time \(t = 0\) this reper takes the referential position: \(D_α, D_β, D_γ\). Since both: \(G^M\) and \(D^a\) are referential and known, it is possible to define a shifter tensor \(S = \delta^M_α D_α G^M\) which describes a connection between both systems: \(G^M = \delta^M_α D_α\). According to Cosserats, we do not lose any generality if we supposing that both referential systems coincide: \(G^M = \delta^M_α D_α\) being a set of orthogonal vectors.

Two fundamental unknown fields of Cosserat continua are: the placement \(x = x_α d_α\) and the proper rotation: \(R = d_α \otimes D^α\), both measured from the intrinsic observer view point. Now, let us define the fundamental for the Cosserat four-time intrinsic formulation two basic measures of continuum velocities, both expressed within the terms of referential gradient:

\[
F = \text{Grad}_α x; \quad F^* = \text{Grad}_d R
\]

(2)

These gradients should be next “transported” into intrinsic frame where, according to the Cartan definition of a connection, they constitute a proper expression for, as Cosserats called, the “geometrical velocities”.

3. Cosserat measures of geometrical velocities

The first geometrical velocities, called by the name “translational geometrical velocities”, defined collectively for zero-, one-, two- and three-dimensional bodies (the so-called the Cosserat D-brane) is defined to be:

\[
V = FR^{-1} = [x \otimes (\partial_i G + \partial_α G^{αβ} + \partial_{αβ} G^i + \partial_i G^{αβ})]R^{-1} =
\]

\[
= [\partial_i x \otimes d_α + \partial_α x \otimes d_α + \partial_{αβ} x \otimes d_α + \partial_α x \otimes d_α] =
\]

\[
= x_α i d_α \otimes d_α + x_α x_β d_α \otimes d_β + \text{(rigid bodies)}
\]

\[
+ x_α x_β d_α \otimes d_β + x_α x_β d_α \otimes d_β + \text{(rods)}
\]

\[
+ x_α x_β x_γ d_α \otimes d_β + x_α x_β x_γ d_α \otimes d_β + \text{(surfaces)}
\]

\[
+ x_α x_β x_γ x_δ d_α \otimes d_β + x_α x_β x_γ x_δ d_α \otimes d_β \text{(3D body)}
\]

(3)
Now, let recall the common notation of translational geometrical velocities, taken from Poisson’s mechanics where: \( \xi, \eta, \zeta \) - for a rigid body; \( \xi_a, \eta_a, \zeta_a \) - for surfaces; \( \xi_a, \eta_a, \zeta_a \) - for 3D body. It means that eq.(2) can be shortly written as follows:

\[
V = FR^3 = \left\{ \begin{array}{ll}
\xi d \otimes d + \eta d \otimes d + \zeta d \otimes d = \nu d_a \otimes d_a \\
\xi d \otimes d_a + \eta d \otimes d_a + \zeta d \otimes d_a = \nu d \otimes d_a \\
\xi d_a \otimes d_a + \eta d_a \otimes d_a + \zeta d_a \otimes d_a = \nu d_a \otimes d_a \\
\end{array} \right.
\]

where accordance is held with a Poisson rigid body dynamics putting \( d = 1 \). It is easy to find that \( V = FR^{-1} \) is a complicated, nonlinear function of the placement \( x \) and the rotation. In particular, the Darboux vectors: \( I_1, I_2, I_3, I_4 \), defined by the relations like: \( d_{ai} = I_1 \times I_2 \); \( i = t, s, \Omega \), according to Poisson’s denotations are: \( I_i = p_i d_i + q_i d_i^2 + r_i d_i^3 \). Further, according to results by Pietraszkiewicz and Eremeyev [1] and Eremeyev [1], and taking into account different denotations of the “rotational geometrical velocities” we can collect different results into a single concise definition:

\[
L = \frac{1}{2} eFR^4 = \left\{ \begin{array}{ll}
I \otimes I = (p_1 d_1 + q_1 d_1^2 + r_1 d_1^3) \otimes (p_1 d_1 + q_1 d_1^2 + r_1 d_1^3) \\
I_1 \otimes I_2 = (p_1 A_1 + q_1 A_1^2 + r_1 A_1^3) \otimes (p_1 A_1 + q_1 A_1^2 + r_1 A_1^3) \\
I_1 \otimes I_3 = (p_1 A_1 + q_1 A_1^2 + r_1 A_1^3) \otimes (p_1 A_1 + q_1 A_1^2 + r_1 A_1^3) \\
I_2 \otimes I_3 = (p_1 A_1 + q_1 A_1^2 + r_1 A_1^3) \otimes (p_1 A_1 + q_1 A_1^2 + r_1 A_1^3) \\
\end{array} \right.
\]

The above formulæ form one of the most marvelous sets in a whole field theory, underlying the main features of the four-time formulation of the local von Helmholtz group of symmetry. The Darboux vectors: \( I_1, I_2, I_3, I_4 \) are the function of rotation only – it should be noted that it is a single formulæ: for a rigid body, rods, surfaces and 3D body - independently of which case is calculated.

4. Intrinsic flux of symmetry

Cosserat brothers have also introduced a concise system of internal measures of momentum and angular momentum fluxes. Independently of the dimension of body (a rigid body, rods, surfaces, 3D body) they proposed the following measures: \( A'_i, B'_i, C'_i; \ i = t, s, \Omega, b \) for translational fluxes of symmetry and: \( P'_i, Q'_i, R'_i \) for rotational fluxes of symmetry. These measures appear in all Cosserat bodies, therefore, we propose to introduce a single, unified, definition – for the translational fluxes:

\[
T_g = JN^{-1} t = \left\{ \begin{array}{ll}
(A d_i + B d_i + C d_i) \otimes d_i = \tau d_i \otimes d_i \\
(A d_i + B d_i + C d_i) \otimes d_i = \tau d_i \otimes d_i \\
(A d_i + B d_i + C d_i) \otimes d_i = \tau d_i \otimes d_i \\
\end{array} \right.
\]

and for rotational fluxes

\[
M_g = JN^{-1} m = \left\{ \begin{array}{ll}
(P d_i + Q d_i + R d_i) \otimes d_i = \mu d_i \otimes d_i \\
(P d_i + Q d_i + R d_i) \otimes d_i = \mu d_i \otimes d_i \\
(P d_i + Q d_i + R d_i) \otimes d_i = \mu d_i \otimes d_i \\
\end{array} \right.
\]

Above, \( T = det F \), and the nonsymmetrical momentum flux \( t = t'' g_g \otimes g_t \), the angular momentum flux \( m = m'' g_g \otimes g_t \) which are usually used for momentum and angular momentums balances. The above definitions of symmetry fluxes are independent of dimension of Cosserat continua, in the literature, are related to as the intrinsic formulation. Generally, the translational measures: \( v_{ai}, r_{ai} \) as well as the rotational one: \( l_{ai}, \mu_{ai} \), \( i = t, s, \Omega, b \), are well known within the dynamics of a rigid body, rods, surfaces and 3D body [2]. Especially, well known is a similarity between equations of the rigid body dynamics and rods statics, in the literature given a name of the “Kirchhoff analogy”.

5. Conclusions

The four-time unification concept, proposed by Cosserats more than one hundred years ago, now is intensively developed within the framework of the quantum field theory. It is surprising that superstring theory is not just theories of one-dimensional objects. There are higher dimensional objects with dimensions from zero (points) to nine – such objects are called p-branes. In terms of branes, we usually call a membrane would be a two-brane, a string is called a one-brane and a point is called a zero-brane. It is easily to find a precise analogy with the Cosserat-branes if we remember, that string lives in ten-time dimensions, which means one real time dimension plus nine time-like dimensions. A special class of p-branes in string theory is called D-branes. Commonly speaking, a D-branes is a p-brane where the ends of open strings are localized on the brane.

References


The creep behaviour of high-temperature rotating components with power-law constitutive models

Mariusz Banaszkiewicz
Energy Conversion Department, The Szewalski Institute of Fluid-Flow Machinery PASc
Fiszera 14, 80-952 Gdańsk, Poland
e-mail: mbanaszkiewicz@imp.gda.pl
Thermal Services, ALSTOM Power Ltd in Warsaw, Branch Elblag
Stocznia 2, 82-300 Elblag, Poland

Abstract

The paper discusses the application of a Characteristic Strain Model (CSM) to the analysis of creep behaviour of rotating components. Simple cylinders are analysed at variable loads and different model constants. Hollow cylinder behaviour is investigated by numerical analysis and the skeletal point location shown to be independent of the applied load. Finally, a numerical creep analysis of a steam turbine rotor is carried out with a detailed examination of stress fields in the rotor disc. The existence of multiple skeletal points in the rotor disc is shown, as well as independence of their location of the creep data used.

Keywords: creep, high-temperature creep model, steam turbine rotor

1. Introduction

Over the past decades, considerable development has been made in understanding creep mechanism, its modelling and defining design methods for creep. A comprehensive review of creep analysis and design methods was presented by Yao et al. [6] who classified creep models in three groups: classical plastic theory, cavity growth mechanisms and continuum damage mechanics based models. They concluded that the existing creep models or constitutive equations are sensitive to material and temperature, and due to this a general model or constitutive equation is yet unavailable.

Holdsworth et al. [4] emphasized that from practical point of view it is important for the creep model to be simple and effective in description of creep deformation at long times. An example of such a model is a characteristic strain model developed by Bolton [2] which is an extension of the Norton model for a steady state creep [5].

The paper presents the results of creep analysis of rotating components subjected to volumetric centrifugal load using the characteristic strain model. The analysis was carried out for simple cylinders and for a steam turbine rotor representing a more complicated component with notches.

2. Power-law creep models

In the most widely used Norton model for a steady state creep, the minimum creep rate is described by a power law relationship [5]:

\[ \dot{\varepsilon}_c = B\sigma^n \] (1)

where \( \sigma \) is constant stress, and \( B \) and \( n \) are constants determined at given temperature. This relationship expresses a constant slope of the log-log curve of strain rate or strain and stress, with the slope being the power exponent \( n \). Experimental creep curves for creep-resistant steels plotted in \( \log(\varepsilon) - \log(\sigma) \) coordinates show that the slope is not constant, \( n \) is constant, and at small strains and stresses it is close to unity, while at high strains, when the stress is a large fraction of the creep rupture strength, the exponent \( n \) tends to infinity.

Based on the analyses of creep tests data Bolton proposed the following isochronous relation between stress exponent and stress expressed as a fraction of creep rupture strength \( \sigma \) [2]:

\[ \frac{\partial \log \varepsilon_c}{\partial \log \sigma} = n = \frac{1}{1 - \sigma/\sigma_c} \] (2)

where \( \sigma \) denotes the creep rupture strength for a given time and at constant temperature.

Integrating the above relationship, Bolton obtained an equation for creep strain \( \varepsilon_c \) in the form:

\[ \varepsilon_c = \frac{\varepsilon_f}{\sigma_f/\sigma_c - 1} \] (3)

where \( \varepsilon_f \) is a characteristic creep strain being a material constant at a given time and temperature. The characteristic creep strain can be evaluated using Eqn (3) and two values: the creep rupture strength \( \sigma_{R1} \) at time \( t_i \) and the stress \( \sigma_{R2} \) to produce datum creep strain \( \varepsilon_d \) at time \( t_i \). With this assumption, the isochronous stress-strain relation of Eqn (3) can be written

\[ \varepsilon_c = \frac{\varepsilon_f (\sigma_{R1}/\sigma_{R2} - 1)}{(\sigma_{R1}/\sigma_c - 1)} \] (4)

Assuming a power-law relationship for the rupture strength, the model relationship between creep strain and time at constant stress assumes the form:

\[ \varepsilon_c = \varepsilon_f (\sigma_{R1}/\sigma_{R2} - 1) \left[ \left( \frac{\sigma_{R1}}{\sigma} \right)^{\frac{1}{m}} - 1 \right] \] (5)

where \( m \) is exponent in the power-law relationship for the creep rupture strength.

3. Analysis of rotating cylinders

A solution of equilibrium equation obtained by Chmielniak et al. [3] for a rotating solid cylinder at creep described by the Norton model shows that the stress components are independent of the creep model parameters, and the equivalent Huber-Mises stress calculated based on these stress components is constant and equal:

\[ \sigma_{eq} = \frac{\rho \omega^2 r^2}{4} \] (6)
Table 1: Creep strengths at 500°C

<table>
<thead>
<tr>
<th></th>
<th>Rupture strength at 10⁵ hrs</th>
<th>Rupture strength at 2x10⁵ hrs</th>
<th>0.2% creep strength at 10⁵ hrs</th>
<th>Characteristic strain</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>[MPa]</td>
<td>[MPa]</td>
<td>[MPa]</td>
<td>[%]</td>
</tr>
<tr>
<td>Min</td>
<td>130.0</td>
<td>105.0</td>
<td>40.3</td>
<td>0.4452</td>
</tr>
<tr>
<td>Mean</td>
<td>162.5</td>
<td>131.2</td>
<td>50.4</td>
<td>0.4448</td>
</tr>
<tr>
<td>Max</td>
<td>195.0</td>
<td>157.5</td>
<td>60.5</td>
<td>0.4460</td>
</tr>
</tbody>
</table>

A similar solution is obtained using the characteristic strain model for creep [1] which shows that stress distributions at steady state creep depend on the rotational speed only (Fig. 1). For the analysed cylinder of outer radius \( r = 0.3 \) m, a skeletal point at \( r = 0.173 \) m is seen, that is located closer to the outer radius.

![Figure 1: Elastic and relaxed stress distributions in solid cylinder at different loads.](image)

In case of a hollow rotating cylinder, both the Norton model [3] and CSM [1] predict stress distributions depending on the rotational speed and on the creep model parameters. With the increasing temperature, accompanied by decreasing creep rupture strength and increasing characteristic strain, the steady-state stress fields become more uniform.

### 4. Creep of steam turbine rotors

Steam turbine rotors operate at non-uniform temperature and stress fields with a high stress concentration. Stress relaxation and creep strain accumulation take place at different rates depending on the location. The effect of creep model coefficients on the stress distribution was investigated varying the creep rupture strength and maintaining the characteristic strain constant [1]. This was achieved using minimum, mean and maximum creep property data shown in Table 1.

The equivalent stress distributions at steady state in the rotor disc radial section obtained with the above data are presented in Fig. 2. The stress distributions calculated with different creep data, corresponding to constant characteristic strain of 0.445% and typical scatter in creep rupture strength of ±20%, are nearly coincident. Approximately 90% of the considered section remains unaffected by the varying creep properties, and in the remaining part the difference in stress between minimum and maximum data is about 10%. Stress relaxation in this region is more significant for minimum data, while the stress curves corresponding to maximum and mean data practically coincide with each other.

Also in the axial section, some skeletal points were found at which Huber-Mises stress remains constant during stress relaxation [1]. The skeletal points are located symmetrically with respect to the disc centreline and closer to the disc side faces.

![Figure 2: Huber-Mises stress distributions across disc radial section at different times.](image)

### 5. Summary

The behaviour of cylinders analysed using the characteristic strain model is similar to that based on the Norton model. In solid rotating cylinder, the stress distributions are independent of the coefficients of creep model equation used, and the equivalent Huber-Mises stress at steady-state creep is constant through cylinder section and depends on the load only. In a hollow rotating cylinder, all stress distributions at steady-state depend both on the applied load and the creep model constants.

For both cylinders the existence of skeletal point was confirmed, the location where elastic and relaxed stress values are the same.

The analysis of steam turbine rotor disc, where two dimensional stress state persists, showed the existence of multiple skeletal points: one was found in the disc radial section and two points found in the axial section. The existence of two skeletal points is a result of symmetric stress field in the disc.

### References


Dynamic substructuring approach for human induced vibration of a suspension footbridge

Bartłomiej Błachowski1, Witold Gutkowski2, Piotr Wiśniewski3*

1,3 Institute of Fundamental Technological Research, Polish Academy of Sciences
Pawińskiego 5b, 02-106 Warsaw, Poland
e-mail: bblach@ippt.gov.pl, pwisn@ippt.gov.pl

2 Institute of Mechanized Construction and Rock Mining
Racjonalizacji 6/8, 02-673 Warsaw, Poland
e-mail: wgutkow@ippt.gov.pl

Abstract

A substructuring method for the prediction of the dynamic response of footbridges subjected to loadings induced by a pedestrian is presented. The dynamic system is composed of two independent subsystems. The first one is the model of the suspension footbridge, and the second one is the model of a pedestrian. The former is obtained using the standard finite element method, while the latter is created by applying a two-step identification approach. The first step is an inverse kinematics based on an experimental human motion analysis. The second one relies on Proper Orthogonal Decomposition extracting a number of most important modes, describing the motion of the pedestrian. The presented methodology will be demonstrated by a numerical example of the pedestrian-footbridge interaction.

Keywords: dynamic substructuring, human-structure interaction, suspension footbridges

1. Introduction

Synchronized activity of people can cause a number of dynamical structural problems. One example is a group of humans jumping or dancing on steel-concrete composite floors [1]. Another example is the dynamic interaction of a walking pedestrian with a flexible footbridge. Recently, a number of papers have been devoted to this topic [2,3]. The authors of these papers are using different techniques for analysing the problem. Some of them are using the discrete element method to model crowd motion. However, it seems that a better approach should be based on dynamic substructuring, for instance proposed by Biondi et al. in the case of a train-track-bridge system [4].

In the paper we are following Biondi’s approach however, instead of modelling the pedestrian motion, we are using experimental data to identify it. Then, FE model of the suspension footbridge and the identified model of the pedestrian will be coupled using the dynamic substructuring technique.

2. Substructures of the dynamical model

2.1. Substructure 1 – suspension footbridge

Dynamic data of an existing a 67.1m long suspension footbridge are used for the numerical simulation. A general view of the bridge is shown in Figure 1. The main structural components of the footbridge are: two towers, two cables, 42 hangers, and a suspended deck. The towers are built of two I-beam columns; the tower columns are braced by I-beams. The suspended deck consists of two C-shaped channel stringers, rectangular hollow section floor beams (transoms) and L-shaped bracings (diagonals). The main cable is a stranded wire, and the vertical hangers are placed at 3 m intervals.

![Figure 1: FE model of the suspension footbridge under investigation.](image)

Figure 1: FE model of the suspension footbridge under investigation.

Equations of motion of the suspension footbridge take the form:

\[
M_b \ddot{u}_b(t) + C_b \dot{u}_b(t) + K_b u_b(t) = f_b(t) + g_b(t)
\]

where

- \(M_b, C_b, K_b\) are mass, damping and stiffness matrices, respectively,
- \(u_b(t), \dot{u}_b(t), u_b(t)\) - acceleration, velocity and displacement vectors, respectively
- \(f_b(t)\) - external force vector (such as wind pressure)
- \(g_b(t)\) - interaction force caused by the moving human

2.2. Substructure 2 - pedestrian

The second substructure is a walking human, whose motion is modelled with the aid of a two-step identification procedure. In the first step, experimental data of the human walking provided by the OpenSim project [5], are used to perform inverse kinematics (Figure 2). Then, using MATLAB, Proper

*Acknowledgements. The authors wish to express their gratitude for financial support provided by the National Science Centre, grant NN501 0494 40.
Orthogonal Decomposition of the motion is performed to identify the most dominant modes of the walking human.

![Figure 2: Mass distribution within a pedestrian body](image)

Having the modes and knowing the mass distribution of human body, we linearize the dynamic model of the pedestrian:

\[
M_p \ddot{u}_p(t) + C_p \dot{u}_p(t) + K_p u_p(t) = f_p(t) + g_p(t)
\]

(2)

where left hand side components of the equations (2) are analogous to equations (1), \( f_p(t) \) denotes an external (independent of the footbridge movement) force acting on the pedestrian, and \( g_p(t) \) is the interaction force, which is in equilibrium with \( g_b(t) \).

### 2.3. Pedestrian-footbridge combined model

Finally, dynamic model of both substructures, may be combined into one dynamic system

\[
\begin{bmatrix}
M_b & 0 & \ddot{u}_b(t) \\
0 & M_p & \ddot{u}_p(t) \\
K_b & 0 & u_b(t) \\
0 & K_p & u_p(t)
\end{bmatrix}
+ \begin{bmatrix}
C_b & 0 & \dot{u}_b(t) \\
0 & C_p & \dot{u}_p(t) \\
0 & 0 & f_b(t) \\
0 & 0 & f_p(t)
\end{bmatrix}
+ \begin{bmatrix}
\dot{g}_b(t) \\
\dot{g}_p(t)
\end{bmatrix}
= \begin{bmatrix}
f_b(t) \\
f_p(t)
\end{bmatrix}
\]

(3)

or in a more compact form

\[
\begin{bmatrix}
M_b & C_b & \dot{u}_b(t) + \dot{g}_b(t) \\
0 & M_p & \dot{u}_p(t) + \dot{g}_p(t) \\
K_b & 0 & u_b(t) + g_b(t) \\
0 & K_p & u_p(t) + g_p(t)
\end{bmatrix}
= \begin{bmatrix}
f_b(t) + \dot{g}_b(t) \\
f_p(t) + \dot{g}_p(t)
\end{bmatrix}
\]

(4)

The above equations have to be considered together with the compatibility condition and interface force equilibrium.

\[
\begin{align*}
B(t)u(t) &= 0 \\
L(t)^Tg(t) &= 0
\end{align*}
\]

(5)

where \( B(t) \) is a matrix containing elements dependent on the current position of the pedestrian on the footbridge and the assumed shape functions of the FE model of the footbridge, and \( L(t) \) matrix satisfies the following condition

\[
L(t) = \text{null}(B(t))
\]

(6)

Then assuming a set of independent generalized coordinates we can write

\[
u(t) = L(t)q(t)
\]

(7)

Finally, substituting (6) into (4) and (5) we get

\[
M(t)\ddot{q}(t) + C(t)\dot{q}(t) + K(t)q(t) = f(t)
\]

(8)

The resulting dynamical system is then a linear system containing mass, damping and stiffness matrices varying with time.

### 3. Numerical simulation

The theoretical considerations, presented in the paper, are illustrated with a numerical simulation of the dynamic interaction between the pedestrian and suspension footbridge. The results of numerical simulations are verified using experimental data in the form of acceleration signals from sensors mounted on the cables of the footbridge and on the pedestrian’s back (Figure 3).

![Figure 3: Accelerations of the middle point of the suspension cable](image)

### References


Absolutely unstable round hot jet – a numerical study

Andrzej Bogusławski*, Artur Tyliszczak*, Karol Wawrzak*

1,2,3 Institute of Thermal Machinery, Faculty of Mechanical Engineering and Computer Sciences, Częstochowa University of Technology
Armi Krajowej 21, 42-201 Częstochowa, Poland
e-mail: abogus@ime.pcz.czest.pl

Abstract

The paper presents LES (Large Eddy Simulation) of absolute instability of a round hot jet. It was observed in low density round jets that below a certain critical value of the ratio of the jet density to the density of ambient fluid strong periodic oscillations appear. These oscillations are manifested by axi-symmetric vortical structures generated in the near jet field. Linear spatio-temporal stability theory relates these type of oscillations with the absolute instability characterised by the disturbance development in time. If in sufficiently large region of the jet, the velocity and density profiles support absolute instability development, the so-called global jet instability is observed. The existence of the global instability was verified experimentally in air-heated jets and helium-air mixtures. However, there are still open questions concerning for example critical density ratio that was measured at different levels in hot and air-helium jets. Moreover, some additional oscillations present in variable density jets were observed not justified on the theoretical grounds so far. The study was aimed at numerical simulations of absolutely/globally unstable jet for better understanding of mechanisms leading to this type of instability.

Keywords: absolute instability, variable density jets, Large Eddy Simulation

1. Introduction

Absolute instability of round low-density jets was studied with spatio-temporal linear stability theory by Monkewitz and Sohn [10] and later by Jendoubi and Strykowski [5]. They showed, using Briggs [2] and Bers [1] criterion, that in parallel axi-symmetric low-density jet an absolutely unstable mode, growing exponentially at the location of its generation, can be triggered. The results stemming from the linear stability theory were confirmed in two fundamental experimental works on heated jets by Monkewitz et al. [9] and air-helium jets by Kyle and Sreenivasan [6]. In both experiments strong oscillations were observed for low-density jets. In the case of heated jets [9] the oscillations identified as the absolutely unstable mode were observed for the density ratios lower than the critical one $S_c \approx 0.65$. These oscillations are called Mode II. In the case of air-helium jets the critical density ratio, below which oscillating mode emerged, established by Kyle and Sreenivasan [6], was slightly lower $S_c \approx 0.61$. In both experiments axi-symmetric vortical structures undergoing vortex pairing were observed. Characteristic frequencies of experimentally observed oscillations agreed very well with the results of linear stability theory. However, in both experiments some differences were also indicated. In the case of heated jets additional oscillations, called Mode I, were measured for density ratio $S < 0.69$, while air-helium jets revealed broadband oscillations for very thin shear layer. The origin of these two types of oscillations in low-density jets is not understood up to now.

The LES and/or DNS could bring new insight into understanding of low-density jets transition mechanisms. However, there are surprisingly few numerical studies on variable density jets available in the literature so far. LES predictions for variable-density jets were performed recently by Zhou et al. [15], Tyliszczak and Bogusławski [12], Tyliszczak et al.[13]. These LES results did not show clear presence of absolutely unstable mode which could be compared with the experimental results. DNS predictions of low density jets with wide range of density ratios and shear layer thicknesses were recently performed by Lesshafft et al. [8] for jet at Reynolds number $Re_0 = 7500$. The frequency predictions of the DNS results were substantially higher than those found by linear theory. Recently LES predictions of global mode in round low-density jet at $Re_0 = 7000$ were presented by Foysi et al. [3] for density ratio $S = 0.14$ and shear layer thickness $D/\theta = 27$.

The paper is aimed at LES predictions, similar to those presented by Foysi et al. [3], $Re_0 = 7000$, but for a wider range of density ratio.

2. Numerical method

The flow solver used in this work is an academic high-order code based on the low Mach number approximation. This code (SAILOR) may be used for solving a wide range of flows under various conditions, varying from isothermal and constant density to situations with considerable density and temperature variations. For research purposes the SAILOR code includes a variety of sub-grid models used when the code is operated in Large Eddy simulation (LES) mode [4,11]. In the present work we incorporate the sub-grid model as proposed by Vreman [14].

The solution algorithm is based on a projection method with time integration performed by a predictor-corrector (Adams-Bashforth/Adams-Moulton) method. The spatial discretization is based on 6th order compact differencing developed for half-staggered meshes [7]. Unlike in the fully staggered approach the velocity nodes are common for all three velocity components whereas the pressure nodes are moved half a grid size from the velocity nodes. This greatly facilitates implementation of the code and is computationally efficient as there is only a small amount of interpolation between the nodes. As shown in [7] the staggering of the pressure nodes is sufficient to ensure a strong velocity-pressure coupling which eliminates the pressure oscillations occurring on collocated meshes.

*The research project was supported by Polish National Science Centre, project no. DEC-2011/03/B/ST8/06401 and statutory funds BS/PB-1-103-3010/11/P. This research was supported in part by PL-Grid Infrastructure.
3. LES predictions of absolutely unstable jet

Fig. 1 presents the non-dimensional frequency based on the jet diameter and maximum velocity at the nozzle exit ($S_{\text{th}} = D/\beta U_{\text{max}}$) of the global mode predicted by LES compared to the results of linear spatio-temporal stability and LES results of Foisyi et al. [3] and DNS predictions of Lesshaft et al. [8]. The present LES results for thicker shear layer ($D/\theta = 27$) coincide very well with the DNS of Lesshaft et al. [8]. The global frequency predicted with LES is substantially higher than the absolute mode frequency obtained from the linear stability theory. The discrepancies between LES predictions and stability calculations are increasing for lower density ratios. Present LES prediction of the global mode frequency for the density ratio $S = 0.14$ differs also from the results of Foisyi et al. [3].

Figure 1: Global frequency predicted with LES and DNS compared with absolute frequency of the inlet

Figure 2: Spectral distribution of axial velocity fluctuations at the jet axis and distance $x/D = 3$ from the nozzle exit

Figure 2 presents sample spectra of the axial velocity fluctuations registered at the jet axis and distance $x/D = 3$ from the nozzle exit for the density ratio varying in the range $S = 0.2 \pm 0.6$ and shear layer thickness $D/\theta = 27$. For the density ratio $S > 0.7$ there are no visible periodic oscillations, while strong peak is emerging for the density ratios $S \leq 0.6$.

4. Conclusions

The paper presents LES predictions of global mode in low-density round free jet. The results suggest that the global oscillations were reproduced for wide range of the density ratio below the critical value. The critical density ratio was predicted with reasonable agreement with the results of spatio-temporal linear stability theory. However, characteristic frequencies of the global mode are overpredicted compared to the stability calculation results. The present LES results are in good agreement with the DNS reported by Lesshaft et al. [8], while some discrepancies were observed with the LES predictions of Foisyi et al.[3].

References

Analysis approach for a diffusor augmented small wind turbine rotor

Jakub Bukala¹, Krzysztof Damaziak¹, Krzysztof Kroszczyński³, Marcin Krzeszowiec¹, Jerzy Malachowski², Krzysztof Sobczak⁶

¹,²,³,⁴,⁵ Department of Mechanics and Applied Computer Science, Military University of Technology
Kaliskiego 2, 00-908 Warszawa, Poland
e-mail: jbukala@wat.edu.pl, kdamaziak@wat.edu.pl, mkrzeszowiec@wat.edu.pl, jerzy.malachowski@wat.edu.pl
⁶Institute of Turbomachinery, Lodz University of Technology
Wólczańska 219/223, 90-924 Łódź, Poland
e-mail: krzysztof.sobczak@g.p.lodz.pl

Abstract

This paper presents a proposed workflow for initial design and simulation of a small diffusor augmented wind turbine rotor using modern computer methods. The authors describe the methodology behind the process and cover subjects of numerical wind data acquisition, establishing blade geometry using a 2D porous body approach and simulating that geometry in a series of 3D, 2-way fluid-structure interaction CFD simulations.

Keywords: small wind turbine, diffusor augmented turbine, renewable energy, wind power, fluid-structure interaction

1. Introduction

Increasing interest in renewable energy sources, due to country level tax incentives and rising oil prices, is driving research towards developing a range of advanced new energy generation devices and unit solutions. Wind power is a good example in this field. As a direct result of rising understanding of climate change and negative aspects of green gas emission there is an increasing number of people taking interest in setting up private small wind power plants. As opposed to large wind turbines, dominated by the classic three-bladed horizontal axis setup [1], the small turbine market offers a number of different solutions, often advertised more efficiently than the popular three-blade design. An in-depth analysis of the European market of small wind turbines has performed by various authors, with limited insight on the features of the turbine design [2,3]. An interesting design solution for a small wind turbine with a horizontal axis of rotation is the Diffusor Augmented Wind Turbine (DAWT). The design was firstly patented and built over 150 years ago by Ernest Bolée. After the decades of stagnation in the wind turbine market, the design has seen a major interest increase in recent years with many academic [4,5] and industrial centres proposing various solutions to the DAWT concept. The main advantage of such a turbine is the possibility to use smaller size rotor blade, which, in turn, greatly reduces the overall turbine cost, as the blades are expensive parts [1]. Because of the possible cost reduction, the authors have chosen the DAWT concept as a design candidate for a new small wind turbine with the intent of private ownership and micro - energy generation.

2. Wind statistics

The initial task of establishing the turbine parameters was based on a broad statistical study. In order to analysis, accurate wind data with a high enough sample rate was required the
4. **CFD aerodynamic and FEM strength analysis of the blade**

Based on the established blade geometry, a more sophisticated 3D flow model was created. The results of a series of simulations confirmed the power augmenting possibilities of the planned diffusor shroud. Results predict an increase of wind velocity as high as 50% in some areas, in regard to the initial 12 m/s, free flow wind speed. This speed-up effect can be seen on Fig. 2, which presents the wind velocity as a function of the distance from the hub in the radial direction, inside the diffusor and before the flow stream interacts with the rotating blade.

![Figure 2: Sample results presenting flow acceleration from the 3D flow simulation of the shrouded rotor](image)

The blade model was also used to prepare an initial, steady-state strength FEM analysis with regard only to inertia loads. The next step was to establish a coupled system. The pressure map extracted from the CFD simulation was imposed as a load boundary condition for the FEM strength analysis together with the inertia loading. Various materials were tested, ranging from steel and aluminium to carbon and glass fibre composites with various layer layups. Currently, a two-way transient FSI model is investigated, where the CFD pressure maps will be tightly coupled with the FEM displacement maps with regard to a changing time. The final results from this model will be presented at the conference and in a later paper.

5. **Conclusions**

The method presented in the paper provides a robust and efficient approach to designing new small wind turbines. It is based on a natural synthesis of various advanced computer methods such as weather forecasting, statistical analysis, CFD and FEM simulations. Such a broad range of tools allows for an early identification of the most important turbine parameters and incorporating numerous findings into the proposed design with relative ease. The work is still in progress, but actually the authors conclude the following:

- Weather prediction modelling can be used successfully for turbine parameter response analysis.
- The use of a shrouded turbine can reduce the rotor diameter by as much as 60%, while at the same time retaining the power output of a non-shrouded rotor.
- A well-designed small wind turbine can be economically justified for land mean wind speeds of over 5 m/s, as long as the initial investment is not very costly and the supporting tower is at least 15 m high, preferably higher.
- At this stage of the research glass fibre composite materials for the blades of a small wind turbine appear to be the most versatile and effective.

**References**


Geometric analysis of a 1-DOF, six-link feeder

Jacek Buśkiewicz
Faculty of Mechanical Engineering and Management, Poznań University of Technology
Piotrowo 3, 60-965 Poznań, Poland
e-mail: jacek.buskiewicz@put.poznan.pl

Abstract

The paper deals with the geometric analysis of a one degree of freedom feeder built up of six links. The proposed structure does not require any extra drive of the gripper as the jaws of the gripper are driven by the same active link which drives the whole feeder. The jaws catch the product and transport to other work stand where the gripper releases the product and moves back to its initial position. There exist various different links assemblies for which there exist phases of the gripper closed and open, but they result in mechanisms differing in kinematic parameters. The analysis is carried out to ensure that the active link makes a full revolution and that the area enclosing the trajectory of the gripper is of a reasonable size.

Keywords: six-bar linkages, mechanism analysis, feeder

1. Introduction

The proposed structure of a feeder is based on kinematic chain in which there exists the phase of motion when two links rotate at the same angular velocity. This effect can be obtained by locking the coupler of the four-bar linkage in the dwell mechanism [5-9]. Then the frame of the dwell mechanism becomes a moving coupler, and the other coupler is the output rotating link in the dwell mechanism. The advantage of the proposed solution is that the motion is transmitted mechanically from the driving link to the gripper and therefore the jaws does not require any additional independent control, steering or other tightening elements as the other known feeders do [3,4]. In order to design a one-degree-of-freedom feeder built up of six links connected by means of revolute joints, the techniques for optimal path/motion synthesis of four link planar mechanisms outlined in [1,2] were employed. Compared to [1,2] the paper presents a detailed analysis carried out to ensure that the active link makes a full revolution and to estimate the working space of the mechanism.

2. Geometry of 1 dof six-link feeder

The main chain of the feeder is the four-bar linkage $O_1ABO_2$ with point D tracing the trajectory a part of which is the circular arc of radius $R$, centered at point $O_3$ and spanned by a given angle (Figure 1a). A crank-rocker linkage is accepted only in order to simplify the drive of the mechanism. Let the mechanism be set in the initial position, in which point D is located at the beginning point of the circular arc. An additional link is fixed to the ground at the pivot $O_3$. The length of the link is $l_7 = R$. We choose an arbitrary point M on coupler AB and we connect the coupler to link MN by means of the revolute joint at this point. The links MN and NO are connected by a revolute joint N. The location of point M must be such that the full revolution of the active link is allowed. The adequate conditions are formulated in order that links O3N and NM do not approach to the position in which they are collinear. The position of joint N is coincident with the position of point D when D moves along circular arc. Then link MN and a coupler AB rotate with approximately the same angular velocity. One jaw of the gripper is attached to coupler AB, while the other one is attached to link MN (Figure 1b). When point D starts drawing the arc, the jaws of the gripper catch the product, shut and keep closed as long as point D moves along the circular arc. Subsequently, the gripper opens, the product is taken away and point D moves back to its initial position along the remaining part of the coupler curve.

Figure 1: The geometric scheme (a), the general concept (b)

The last step is to choose the location of the characteristic point $G_1$ of the first jaw in the local system $Ax'y'$ (Figure 1b). The location of this point is very important when the path, along which the load is carried, is also prescribed. Theoretically the location of point $G_1$ can be chosen arbitrarily since the feeder can be translated, scaled and rotated so that the load moves between two preset pickup and destination points. Nonetheless, the location of $G_1$ influences on the size of the working area of the feeder and determines whether it is possible to fix the feeder into existing system of machines. Further, may be computed coordinates of the point of the jaw on coupler NM (Figure 1a). The point is denoted by $G_2$ and its coordinates are unequivocally defined in terms of the coordinates of $G_1$. For the sake of computing $G_2$, the local system $Nx''y''$ attached to
a coupler NM with origin at N is introduced. When the jaws of the gripper are closed, the coordinates of both points G1 and G2 in the global immoveable reference frame Ox1y1 are equal. This is expressed by means of the linear equation leading to the coordinates of G2.

It is obvious that there exist different assemblies of links MN and O3N providing the phase when the couplers do not relatively rotate but they give mechanisms differing in kinematic parameters depending on the location of pivot M and gripper G. The analysis has to be carried out to ensure that the position of point M allowing for a full revolution of active link and that the working space of the gripper is of an acceptable size.

3. Numerical analysis and conclusion

The example presents how the locations of points M and G affect the working space of the mechanisms. A following six-bar mechanism performs motion with the phase when the couplers do not rotate with respect to each other: O1(-5,0), O2(5.78,0), O3(0,0), l1 = 2.785, l3 = 3.348, l5 = 5.032, l6 = 5.833, β = 3.124 rad, θ4 = 0.0094 rad. Lengths are nondimensional. The length of link O3N (pinned at O3) is 3. The allowed positions of a joint M in prescribed area are presented in Fig. 2.

Figure 2: Area of allowed locations of joint M in Ax’y’

From this area the following coordinates of M in the local coordinate Ax’y’ system are taken: (-4.346,2.173). The length of link MN is 7.267. The following coordinates are taken for G1(1.63,16.297) on the first jaw. Based then, the coordinates of the point are derived on the other jaw in the reference frame attached to coupler MN are derived: G2(6.911,-15.096). Applicability of the solution is assessed on the basis of the space occupied by the moving feeder. Therefore, the paths of all the joints which influence the size of the housing are presented.

Figure 3: A position of the mechanism while the gripper is closed (the trajectories of G1 and G2 overlaps)

Figure 3 shows the instant when the gripper is closed. The difference between the angular positions of the both links supporting the jaws is less than 0.0004 rad. Let us change the location of joint M on coupler AB, e.g. M(-4.35,2.17). The length of link MN is now 9.62. The positions of G1 and G2 are the same. The kinematic scheme of the feeder obtained along with the trajectories of joints is presented in Figure 4.

The example shows a general feature of the feeder: it occupies a relatively large working space to transport a load between two points. After finding the positions of pivot M ensuring the full revolution of the active link, it may be analyzed how the positions of gripper and pivot M affect the size of the working space in search for a satisfactory solution. The aim of the further studies is to make the method more automatic in searching for feeders optimally satisfying the prescribed requirements.

Figure 4: A position of the feeder for changed location of M

References

Modal approach in the fluid-structure interaction

Wojciech Chajec¹, Adam Dziubiński²*

¹ Materials & Structures Research Center, Institute of Aviation
Al. Krakowska 110/114, 02.256 Warszawa, Poland
e-mail: chajec@ilot.edu.pl

² Computational Fluid Dynamic and Flight Mechanics Group, Center of New Technologies, Institute of Aviation
Al. Krakowska 110/114, 02.256 Warszawa, Poland
e-mail: Adam.Dziubiinski@ilot.edu.pl

Abstract

The paper presents examples of computational analysis based on modal modelling of the structure motion. This approach is typical for investigation of aircraft aeroelastic phenomena. The flutter computation in the frequency domain, using simple unsteady aerodynamic models and dedicated software is nowadays an aerospace standard. The selected normal modes, control surface modes and rigid body modes should be here taken into account during design. The mechanical properties of structure are represented by modal parameters as mode shape, generalized mass, normal frequency and damping coefficient. For selected configuration of structure they can be determined only once, by computation or ground vibration test. Similar approach can be used for more detailed simulations in the time domain, in which a more credible unsteady aerodynamic models can be applied. Due to this approach, the “structural” part of the problem is very simple. The resulting database (structural modes) is small and unvarying, so the whole time-domain computation can be provided only for a flow computation system, without data exchange between structure’s and flow computation systems.

Keywords: modal approach, normal modes, rigid body modes, fluid-structure interaction, frequency domain, time domain

1. Introduction

Modal modelling of the structural motion is a time and cost efficient way of dynamic analysis. The selected normal modes, control surface modes (including control system modes) and rigid body modes (for unsupported structures as flying aircrafts) should be here taken into account.

The rigid body modes of unsupported structures can be regarded as a simple extension of normal modes. The ordinary normal modes (vibration modes) are orthogonal to rigid body modes due to constant momentum equations. The rigid body modes are mutually orthogonal, if they were selected as displacements along and rotation about main central axes of inertia of the structure.

The mechanical properties of structure (it should be close to real structure) are represented by modal parameters as mode shape, generalized mass (unity for a normalized mode shape), normal frequency (zero for rigid body modes) and damping coefficient.

For a selected configuration of a structure, the modes can be determined only once, by computation or ground vibration test. Some alterations of this configuration (as fuel or payload mass determination only once, by computation or ground vibration test. Similar approach can be used for more detailed simulations in the time domain, in which a more credible unsteady aerodynamic models can be applied. Due to this approach, the “structural” part of the problem is very simple. The resulting database (structural modes) is small and unvarying, so the whole time-domain computation can be provided only for a flow computation system, without data exchange between structure’s and flow computation systems.

Using a modal approach, the equations of motion can be written as (each mode shape is normalized with respect to unit generalized mass):

\[ \ddot{q}_i + c_i \dot{q}_i + \omega^2 q_i = f_i \quad i = 1, \ldots, n, \]

where

- \( q_i \) is a mode number (selected normal modes, control surface modes and rigid body modes are taken into account),
- \( f_i \) - generalized aerodynamic forces, determined by fluid dynamics, gyroscopic forces and other external forces.

The solution of Eqs (2) is a column matrix of modal coordinates \( q_i(t) \), \( i = 1, \ldots, n \).

---

This paper contains some results of computation, that were supported by Polish Ministry of Science and Higher Education in 2009-2012 as the research project No. 2224/B/T02/2009/37.
3. Typical additional assumptions

Harmonic motion and linear model of unsteady aerodynamic phenomena are typical assumptions to investigate of aircraft aeroelastic phenomena, such as: aerodynamic flutter, airfoil divergence, control reversal, aeroelastic dynamic response (with external excitation) as well as whirl-flutter of propeller power plants (with gyroscopic forces and aerodynamic forces on rotating propeller). The flutter computation in the frequency domain using simple unsteady aerodynamic models (DLR or even strip model) and home (as the Polish JG2, [1], [3]) or commercial software (as the ZONA software – e.g. [5], NASTRAN – e.g. [4], [5] - or Martin Hollmann’s SAF) is an aerospace industrial standard now.

4. Time-domain computation. Advanced models of flow

The equations (2) can be solved directly, in the time domain. This idea was used by several authors, i.e. in the TAURUS and CESAR European projects, [2]. To determine the aerodynamic forces, the normal modes $\varphi_i(r)$ should be known in the selected points. It is possible due to numerical interpolation procedure.

A simple example of such approach has been done in [5]. An advanced fluid dynamics method to solve Reynolds Averaged Navier Stokes (RANS) equations has been used to simulate the movement of two-body object. An airfoil with control surface (here named also a flap, Figure 1) which has the ability to move in three degrees of freedom: vertical movement, whole body rotation and a flap rotation, has been simulated in time domain. This way the object is prepared to represent a flutter phenomena on a wingtip.

Since this movement is well defined in two-dimensional computational space, a tetrahedral mesh representing 2D geometry has been created and flow has been simulated on a widely recognized commercial FLUENT code. The software has an ability to be extended by parts of the code written by the user (UDF), so the movement of both parts of a body has been described using a code with a modal approach mentioned above.

The movement of a body inside the mesh causes the mesh either to be deflected, or to be partially rebuild. This process could cause a mesh to degenerate, it really did for the cases where the movement of body parts has been large in comparison to the mesh density. Unexpectedly this degeneration did not influence the flow solution quality in a similar way that it affected the mesh. This is due to the effective interpolation procedures, that the above mentioned commercial code uses. The results correspond with the well-known behavior of the simulated phenomena, so there is a boundary velocity below which the oscillations tend to be dampened, but above it, the simulation led to rapid movements of the model.

The simulation was an recreation of wind tunnel experiment that was provided by Lorenc [5]. Sub-reference [5], so a computational domain had to be bounded by the walls at least on the upper and lower surfaces. The authors decided to use the condition, which supposed to be safer in terms of a mesh rebuild and avoiding the possibility of wall collision. The walls of domain were sufficiently far from the model to use a far field boundary condition, as in free flight simulations.

A complication of a fluid model has been investigated too. Neglecting the viscosity, forces at different levels of model (mechanic, fluid) has been tested and caused almost no effect if only the shear force has been neglected during integration over the model as on an inviscid surface, but clearly increased the oscillation frequency when inviscid fluid model has been used.

5. Full paper announcement

During the congress the most interesting results of the frequency-domain and time-domain flutter analysis will be presented, comparing the results by means of a number of analytical methods. The result sensitivity (control system modes and element masses, aerodynamic model – control surfaces hinge moments, etc.) and possible errors will be discussed in correlation with the wind tunnel and in-flight tests results.

References


Free vibrations and buckling stability of micro-nonhomogeneous plate band resting on an elastic subsoil

Marek Chalecki¹, Grzegorz Jemielita²

¹,²Faculty of Civil and Environmental Engineering, Warsaw University of Life Sciences (SGGW), Nowoursynowska 164, Warsaw, Poland
e-mail: marek_chalecki@sggw.pl, g.jemielita@gazeta.pl

Abstract

The aim of this work was to consider vibrations of a micro-heterogeneous FGM plate band, compressed with big axial forces and resting on a heterogeneous elastic subsoil. The band consists of number of cells with functionally graded mechanical properties. The proposed way of the problem solution is based on the displacement method and is alternative to various homogenization techniques, being applied in calculations of heterogeneous media.

Keywords: plate band, natural frequency, buckling, elastic subsoil, displacement method

1. Introduction

The work considers a certain model of a micro-heterogeneous FGM plate band with the heterogeneity along the direction of action of big axial forces (it is assumed that this is the x direction). The plate band rests on a heterogeneous elastic subsoil (Fig. 1). Mechanical properties of the band do not depend on the coordinate in another direction (y). If big axial forces and the subsoil heterogeneity are taken into consideration, it is possible to investigate their influence on the natural frequencies of the band as well as to determine the buckling critical force for the band.

Figure 1. Plate band with microstructure

The assumption that the band thickness (h) is constant and that the material properties of the basic cell (Fig. 2) are jumps-type variable along the x axis enables to determine the natural frequencies of such band in an exact way with the use of the equations of the theory of thin plates. A basic plate band (cell) shown in Fig. 2 consists of three bands (subcells) having two different features: width (length) l₁, l₂, stiffness D₁, D₂ and mass per area unit μ₁, μ₂. Due to this, the bands with the features: l₀, μ₀, D₀ (α = 1, 2) are described by two differential equations. If the stiffness and mass density of the band as well as the subsoil stiffness and boundary conditions are independent on the y variable and each of the three parts of the cell (Fig. 2), resting on a subsoil of constant characteristic, has a constant stiffness and mass per area unit, then the differential equation of free vibrations of such part α can be presented in the form

\[
d^4w(x, y) + 2\sigma^2 \frac{d^2w(x, y)}{dx^2} \frac{d^2w(x, y)}{dy^2} - \lambda^2 w(x, y) = 0
\]

where

\[
a^2 = \frac{SI_0}{2D_0}, \quad \lambda^4 = k_{Ba} - k_{Ba} = \frac{\mu_0 h^3\omega^2}{D_0} - \frac{k_{Ba}^2}{D_0},
\]

D₀ = \frac{E_0 h^3}{12(1 - \nu^2)}

the subscript α = 1, 2, \xi_α = l₀/l, S – a constant compressing force, k₀ – the subsoil stiffness under the part α, α – the band natural frequency being investigated.

The paper shows that it is possible to obtain the exact solution of the mentioned problem with the use of the methods of structural mechanics. So far such problems have been investigated with homogenization techniques (eg. the tolerance averaging – cf. [2]) whereas the methods of structural mechanics (displacement method) were applied in [1] to investigate only free vibrations of a micro-heterogeneous plate band.

The aim of the work is to determine the natural frequencies and/or the buckling critical force in an exact way with consideration of the subsoil elasticity for the thin plate band with the microstructure shown in Fig. 1.

It must be mentioned that for the investigations of the influence of big compressing forces on the vibrations of micro-periodic plate band and for calculations of critical buckling forces the Kirchhoff model of thin plate is definitely better than the model of medium thickness plate (Reissner, Ulfstand, Hencky-Bolle, Mindlin etc.). The influence of big axial forces is much visible for thin plates than for medium thickness plates.

2. Basic assumptions

The band spreading along the x axis consists of N cells with the constant length l = L/N. Each cell consists of three subcells: the two outermost ones have the stiffness D₁ and mass per area unit μ₁, the middle one has the stiffness and mass per area unit D₂, μ₂ respectively, wherein each of these subcells rests on the subsoil with the stiffness k₁, k₂ respectively (Fig. 2). Moreover, it is assumed that
\[ D_1 = \frac{E_h^3}{12(1-\nu^2)} \quad \text{and} \quad D_2 = \eta D, \quad D > 0, \]
\[ \mu_1 = \mu = \rho h, \quad \mu_2 = r_\mu, \quad \mu > 0, \quad k_1 = k, \quad k_2 = pk, \]
\[ \text{(3)} \]

where \( \eta \) – a real positive number, \( r, p \) – real numbers equal or greater than 0. It is evident that the band cell can be regarded as a bar element with the section \( F = h \times b, b = 1 \text{ m} \) – in such case \( \mu \) is a mass per length unit [kg/m].

The lengths of the subcells of a \( j \)-th cell are calculated from the formulas
\[ l_1 = \xi_1 d, \quad l_2 = \xi_2 d, \]
where
\[ \xi_1 = \frac{1}{2} \left( 1 - \frac{j - 1}{N + 1} \right), \quad \xi_2 = \frac{j - 1}{N + 1}, \quad j = 1, 2, 3, \ldots, N, \]
\[ l = 2l_1 + l_2 = \frac{L}{N}. \]
\[ \text{(4)} \]

To obtain natural frequencies or a buckling critical force, the stiffness matrix of a typical bar element (cell) of length \( l = 2l_1 + l_2 \) must be determined.

3. Stiffness matrix of a band element

The band cell, fixed on both ends, can be considered a bar element consisting of three elements having a jump-type variable stiffness as well as a mass per length unit and resting on a subsoil of a jump-type variable stiffness. The ends of this element are subjected to the displacements with amplitudes (rotations and relocations) \( \varphi_{0-1}, \varphi_0, \varphi_{w-1}, \varphi_w \) (Fig. 3).

\[ \begin{bmatrix} K_{j-1} & \Phi_{j-1} \end{bmatrix} \begin{bmatrix} \varphi_{j-1} \\ \Phi_{j-1} \end{bmatrix} = \begin{bmatrix} 0 \\ W_j \end{bmatrix}, \quad \begin{bmatrix} 0 \\ W_{j+1} \end{bmatrix} = \begin{bmatrix} K_{j+1} & \Phi_{j+1} \end{bmatrix} \begin{bmatrix} \varphi_{j+1} \\ \Phi_{j+1} \end{bmatrix}, \]
\[ j \text{ is the column vector of the generalized displacements. The terms of the matrix } K \text{ are the function of the investigated values of the natural frequencies depending on the band compressing force, the subcell material density and the parameters of the elastic subsoil.} \]

4. Band stiffness matrix

Figure 4 schematically shows the band section with any number of cells \( N \). The unknown quantities are generalized displacements of the nodes (\( \varphi_j, \varphi_0, \varphi_{w-1}, \varphi_w \)).

\[ \begin{bmatrix} K_{j-1} & \Phi_{j-1} \end{bmatrix} \begin{bmatrix} \varphi_{j-1} \\ \Phi_{j-1} \end{bmatrix} = \begin{bmatrix} 0 \\ W_j \end{bmatrix}, \quad \begin{bmatrix} 0 \\ W_{j+1} \end{bmatrix} = \begin{bmatrix} K_{j+1} & \Phi_{j+1} \end{bmatrix} \begin{bmatrix} \varphi_{j+1} \\ \Phi_{j+1} \end{bmatrix}, \]
\[ j \text{ is the column vector of the generalized displacements.} \]

The stiffness matrix \( K \) and \( \Phi \) is the stiffness matrix of a \( j \)-th element (cell),
\[ \Phi_j = K_j \Phi_j, \]
where \( K_j \) is the stiffness matrix of a \( j \)-th element (cell),
\[ \begin{bmatrix} k_{ij} & k_{ij+1} & k_{ij+2} & k_{ij+3} \\ k_{ij+1} & k_{ij+2} & k_{ij+3} & k_{ij+4} \\ k_{ij+2} & k_{ij+3} & k_{ij+4} & k_{ij+5} \\ k_{ij+3} & k_{ij+4} & k_{ij+5} & k_{ij+6} \end{bmatrix}, \quad \Phi_j = \begin{bmatrix} \varphi_{j-1} \\ \varphi_j \\ \varphi_{j+1} \\ \varphi_{j+2} \end{bmatrix}, \quad \Phi_j = \begin{bmatrix} \varphi_{j-1} \\ \varphi_j \\ \varphi_{j+1} \\ \varphi_{j+2} \end{bmatrix}. \]
\[ \text{(9)} \]

References


Study on the optimization of the reinforced scissor type bridge

Yuki Chikahiro¹, Ichiro Ario², Jan Holnicki-Szulc³, Piotr Pawlowski⁴, Cezary Graczykowski⁵*

¹,² Department of Civil & Environmental Engineering, Hiroshima University
1-4-1 Kagamiyama, Higashi-Hiroshima, 739-7527, Hiroshima, Japan
e-mail: d131584@hiroshima-u.ac.jp

³,⁴,⁵ Institute of Fundamental Technological Research, Polish Academy of Science
ul. Pawinskiego 5B 02-106, Warszawa, Poland
e-mail: holnicki@ippt.pan.pl

Abstract

The world has seen many kinds of natural disasters in recent years. In the view of civil engineering, it is important to consider how to rebuild damaged infrastructures in order to save lives in an emergency situation. To solve these problems, we proposed a new type of deployable bridge - Mobile Bridge™ - which is possible to transport and construct quickly based on the concept of the multi-folding micro-structures. In this paper we suggest a reinforcing method of the Mobile Bridge combined with strut members and prestressing forces in order to adjust stress or displacement after the bridge is deployment. Each sectional area of reinforced member and prestressing force were subjected to optimization by the use of ABAQUS and optimization algorithm in Mathematica. As a result, we succeed to obtain the optimized layout of the reinforced Mobile Bridge and shown its effectiveness.

Keywords: Deployable structure, Scissor type bridge, Section optimization

1. Introduction

In recent years, the world has seen many kinds of natural disasters, which caused many critical situations for residents life by damage of an infrastructure. The rescue time is very important in emergency situations, therefore we have to consider new types of rescue systems and methods for rapid rebuilding of a damaged infrastructure. In order to solve this problem, we proposed a deployable emergency bridge[1], Mobile Bridge™ (MB), based on the concept of the multi-folding micro-structures[2]. The structural form of the MB is similar to a scissor system for its structural form. The design of the MB enables to reduce the construction time on site by deploying the structural frame directly over a damaged bridge or road.

In the previous projects[3], we succeeded to develop a real-scale MB for vehicles. The expansion of the scissor modules with deck boards from the folded state to the final position is shown Fig. 1. The basic scissor module consists of two linear elements joined at a pivot providing a hinge-connection at their centres. In the fully deployed state the two members are in the shape of the character ‘X’ creating a single scissor unit. This basic scissor unit is connected to a next unit by hinges. Such

In general, the scissor mechanisms are mostly applied in architectural field of temporary domes. Their strength and stability are improved arranging the scissor units as geodesic grid, or optimizing sectional area of the scissor components. In the case of emergency bridge, its design must assure construction speed and structural strength in order to provide safe passage for people and vehicles. In the paper, we propose a method of reinforcement of the MB based on additional strut members and prestressing forces, which minimizes stresses or displacements in the deployed state. The sectional areas of strut members and prestressing forces can also be optimized for improving the limit load capacity of the bridge.

2. Optimization methodology

One of optimization tasks considered in this paper is limit load capacity problem, which is defined with constraints imposed on weight, stress and displacement in Equation (1).

\[
\begin{align*}
\text{Maximize} & \quad P_f, \quad \text{s.t.} \quad W < W_t, \quad \sigma < \sigma_t, \quad \delta < \delta_t
\end{align*}
\]

This optimization problem can be solved in many ways. At first, each sectional area of a strut can be optimised in order to

![Figure 1: The experimental MB: (a) Folded state, (b) Deployment, (c) Completed construction](image-url)

Figure 1: The experimental MB: (a) Folded state, (b) Deployment, (c) Completed construction
type of a structure is different from typical truss structures find a layout satisfying the constraint conditions. In a similar because of dominant effects of bending moments.

Acknowledgment. This research was supported for Dr. I. Ario and Mobile Bridge project. We appreciated that all manufacture of experimental bridge was supported by Akashin Corporation in Japan. Moreover, we appreciated that sample of the aluminium materials were offered by Hoshi-kei-kinzoku Industry Co., Ltd. and Sankyo Tateyama, Inc Sankyo Material Co., and the experiments was supported by Japan Construction Method and Machinery Research Institute in Japan.
Figure 2: Flowchart of optimization procedure.

Figure 3: Numerical model of the experimental MB

### 3. Numerical example

The detailed analysis of the model of the MB can be found e.g. in [3]. The bridge in fully deployed state is simply supported and consists of two scissor units. The total length of the span is 7.0m and the height is 2.0m. The sectional properties of the main frame components, which are made using aluminium alloy A6N01, are supported and consists of two scissor units. The total length of the span is 7.0m and the height is 2.0m. The sectional properties of the A6N01 are: $E=62.5\text{GPa}$ and $\sigma_y=180.0\text{MPa}$. The total weight of the MB is approximately 900kg. During field test, the bridge was successfully loaded with vehicle of weight of 14kN.

For the sake of simplicity in the initial optimization problems, all the sections of reinforced members are assumed as rectangular with $A_0=100.0(\text{cm}^2)$, $I_0=833.3(\text{cm}^4)$, $\rho=2.71(\text{g/cm}^3)$. The live load acts in the centre of the bridge at the node $B_2$. The maximum constraints values for weight and yield stress assumed in the analysis are $W=975.6(\text{kg})$ and $\sigma_y=180.0(\text{MPa})$. The limit displacement is defined as $\delta_L=500=14.0(\text{mm})$ referring to Specifications For Highway Bridges in Japan.

We considered four configuration of the bridge. The initial configuration represented original topology of the MB (fig.3 (a)). The configuration C1 (fig.3 (b)) is the starting point for the optimization procedure, which provides configuration C2 (fig.3 (c)). Because designs C1 and C2 are not technically feasible and require complicated assembling of top members, we introduce configuration C3 (fig.3 (d)), in which only vertical elements have to be added after deployment of the bridge. The following table compares maximum load, total weight, maximal stress, displacement, cross sections of the elements for each configuration of the MB.

<table>
<thead>
<tr>
<th>Configuration</th>
<th>Py (kN)</th>
<th>W (kg)</th>
<th>$\sigma$ (MPa)</th>
<th>$\delta$ (mm)</th>
<th>$A_{cm}^t$ (cm$^4$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial</td>
<td>0.8</td>
<td>623.3</td>
<td>20.4</td>
<td>140</td>
<td>100.0</td>
</tr>
<tr>
<td>C1</td>
<td>1632.0</td>
<td>975.6</td>
<td>118.8</td>
<td>140</td>
<td>100.0</td>
</tr>
<tr>
<td>C2</td>
<td>1969.6</td>
<td>974.5</td>
<td>112.9</td>
<td>136</td>
<td>10.6</td>
</tr>
<tr>
<td>C3</td>
<td>959.0</td>
<td>785.9</td>
<td>108.7</td>
<td>140</td>
<td>100.0</td>
</tr>
</tbody>
</table>

Table 1: Numerical results

4. Conclusions

The presented results allow for the following remarks:

- The proposed methodology allows for successful optimization of the MB.
- The proposed solutions provide high maximum loads and satisfy imposed constraints. This high increase of structural stiffness is caused by higher resistance to global bending.

References


Analysis of bending and torsional vibrations of rotors with using perturbation methods

Bogumił Chiliński¹, Radosław Pakowski²

¹,² Faculty of Automotive and Construction Machinery Engineering, Warsaw University of Technology
Narbutta 84, 02-324 Warszawa, Poland

e-mail: bogumil.chilinski@gmail.com ¹, rpakow@simr.pw.edu.pl ²

Abstract

The paper demonstrates analysis of critical state in a rotational system using analytical methods. A simple physical model of elastic rotor was assumed. The elastic shaft was supported on two bearings, heavy disk was embedded on the shaft. The following issues were presented: physical model, its solution and torque disturbances acting on the motion of the system. Introduction outlines of a considered problem, potential opportunities of it are demonstrated. Next, physical and mathematical model of the analyzed object is described. Then a detailed discussion comes on a mathematical model in the form of nonlinear ordinary differential equations proposed earlier. The possibility to solve such a problem and the simplifications used are presented. Furthermore, the influence of used simplifications on the shape of the analyzed problem is demonstrated. Additionally, the possibility of equations solution presented in the paper is discussed. Moreover, a series of interesting properties of the analyzed system of equations are shown based on approximate solutions. The paper is summarized by plans for future work and synthetic conclusions concerning the innovative control method of critical states.

Keywords: rotational system, nonlinear differential system of equation, perturbation method, critical states, analytical solutions

1. Introduction

The importance of nonlinear problems in the entire engineering and machine construction is increasing. Contemporary industry rarely allows for “constructional compromises” in the form of high values of safety factor or simplified design computation process, which lead to overdimensioning of a designed product. Complex computing models are used extensively. The aim of the action is to create light, economical objects, which fulfil structural assumptions possibly neared to the limitations. The products, to have their operation time exceed the assumed durability, are inappropriate among the contemporary trends. In many cases they will be classified as incorrect.

Thus, the linear models of many phenomena, in many cases, become in accurate although there is a clear transparency and a relatively easy way of seeking the solution. A considerable group of structural problems requires considering the nonlinear aspects, although their influence on the system is low.

2. Model

The fundamental problem is the choice of the model for the purposes of a given analysis. Unfortunately, in most cases it does not mean that this problem is simple and unequivocal. The model may be a kind of representation of reality in the language of a given domain (mechanics, electrotechnology, thermodynamics, etc.) That is why its form depends on the shape of problems. Additionally, the assumed level of accuracy acts strongly on the final form of the model. The level of accuracy can be defined as a collection of phenomena considered to be vital in the modelling process.

Obviously, the increase of model accuracy does not lead to proportional increase of accuracy of the results. It happens that the time spent on improving the model and finding its solutions does not correspond to obtained results. That is why, it is necessary to be careful during this process so as not to start describing the reality too precisely. In this case finding the precise model and its solution could be problematic.

In most cases of the analysis of different type of vibrations, the practice indicates that a method of successive approximations is worth using. In the beginning simple models are used so together with the successive solving the discussed problem is more detailed. Such an approach has two basic advantages:

- it allows to split the set of phenomena existing in a given system into basic, less important,
- it allows to solve a given problem on a level of accuracy, a specified referring to solution of previous simplified problem.

Thus, in case of the analysis of bending and torsional oscillations existing in rotors, the starting point is the simplest model in the form presented in Fig 1.

Figure 1: Model of the shaft

This system is represented by the inert disc with a mass of $m$ and a moment of inertia $I$ fixed eccentrically on the elastic shaft within $e$ from the rotation axis.

3. Equation system of dynamics

The system presented in figure (1) (physical model) can be described as second order nonlinear system of differential equation (mathematical model) is:

$$m \ddot{h} - m \cdot e \cdot \sin \phi \cdot \dot{\phi} - m \cdot e \cdot \cos \phi \cdot \dot{\phi}^2 + k(h) \cdot \dot{h} = 0$$

(1)
\( m \cdot \ddot{v} + m \cdot e \cdot \cos \phi \cdot \ddot{\phi} - m \cdot e \cdot \sin \phi \cdot \dot{\phi}^2 + k(\nu) \cdot v = 0 \)  
\( (I + m \cdot e^2) \cdot \ddot{\phi} - m \cdot e \cdot \dot{h} \cdot \sin \phi + m \cdot e \cdot \ddot{v} \cdot \cos \phi = \Delta M(t) \)  

where:

- \( h(t) \) - horizontal displacement of shaft axis at the point of disc mounting,
- \( v(t) \) - vertical displacement of shaft axis at the point of disc mounting,
- \( \phi(t) \) - angular displacement of (rotation) shaft.

The system of Eqns (1-3) is a direct implementation of the second Newton’s laws of dynamics or any formalism of analytical mechanics e.g. Lagrange equations of II kind or D’Alembert’s principle.

4. Solution

The solution is based on Krylov–Bogoliubov method. This is a variant of perturbation method (method of small parameter) based on the assumption of a special form functions expected to be solutions. Finally, it can be written that a solution (4-6) has the following form:

\[ h(t) = a_1(t) \cdot \cos(\omega_1 \cdot t) - \frac{\Omega^2}{\Omega^2 - \omega_1^2} \cdot \cos(\Omega \cdot t) + e^{-}\frac{\omega_1^2 \cdot \omega_2}{2 \cdot ((\Omega - \omega_1)^2 - \omega_2^2)} \cdot \sin(\Omega \cdot t - \omega_1 \cdot t) \]  
\[ v(t) = a_2(t) \cdot \cos(\omega_2 \cdot t) - \frac{\Omega^2}{\Omega^2 - \omega_2^2} \cdot \sin(\Omega \cdot t) + e^{-}\frac{\omega_1^2 \cdot \omega_2}{2 \cdot ((\Omega - \omega_1)^2 - \omega_2^2)} \cdot \sin(\Omega \cdot t - \omega_1 \cdot t) \]  
\[ \theta(t) = a_3(t) \cdot \cos(\omega_3 \cdot t) + e^{-}\frac{m \cdot a}{2 \cdot I \cdot ((\Omega - \omega_1)^2 - \omega_2^2)} \cdot \cos(\Omega \cdot t - \omega_3 \cdot t) + e^{-}\frac{m \cdot a}{2 \cdot I \cdot ((\Omega - \omega_1)^2 - \omega_2^2)} \cdot \sin(\Omega \cdot t - \omega_3 \cdot t) \]

5. Conclusion

Based on a mathematical model, analytical and numerical solutions were found. The results of Krylov–Bogoliubov method were presented in the paper. Moreover, numerical simulations were done, too. This method is effective to solve the problems of dynamics of certain class of rotor systems.

References

First ply failure FEA of laminated shells undergoing large displacements - 6 parameter shell theory approach

Jacek Chróścielewski1, Wojciech Witkowski2, Bartosz Sobczyk3, Agnieszka Sabik4
1,2,3,4 Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: jchrost@pg.gda.pl1, wojwit@pg.gda.pl2, barsobcz@pg.gda.pl3, agsa@pg.gda.pl4

Abstract

This work presents a modification of Tsai-Wu failure criterion, which meets requirements of the 6-parameter nonlinear shell theory, with asymmetric strain tensor measures under particular constitutive relation. The constitutive relation is an extension of classical approach for 2D nonpolar orthotropic linearly elastic continuum used in 5-parameter shell theory onto the 6-parameter formulation. Presented criterion is used in numerical example of compression-loaded flat panel. The obtained results are compared with numerical and experimental reference solutions.

Keywords: nonlinear 6 parameter shell theory, laminated shells, equivalent single layer, first ply failure, Tsai-Wu criterion

1. Introduction

Nowadays laminated shells are more often used in civil engineering. As typical thin-walled structures, they carry loads primarily by membrane action, being frequently exposed to significant compressive loads, which can lead to instability effects. On the other hand, their load carrying capacity is strongly dependent on the inter or intra failure mechanisms in the laminate itself. Some of civil engineering applications of composite materials do not allow for a certain type of damage i.e. cracks. Therefore, the first ply failure (FPF) method for estimation of the onset of damage, basing on some failure theories, becomes highly important. In the work, the authors present some selected numerical examples concerning estimation of FFP of laminated plates and shells undergoing large displacements, under formalism of Finite Element Method (FEM) and nonlinear 6-parameter shell theory with asymmetric strain tensor measures and drilling rotation.

2. Theoretical approach

2.1. The structure response

In order to describe structural response of thin laminated shell FEM is used within the framework of 6-parameter nonlinear shell theory, with asymmetric strain tensor measures [1]. The Equivalent Single Layer theory (see for instance [2]) is adopted as the representation of multilayered medium. The constitutive relation used in calculations is a straightforward extension of classical approach for 2D nonpolar orthotropic linearly elastic continuum, used in 5-parameter shell theory onto the 6-parameter formulation [2]. It utilises five material engineering constants of the classical elastic continuum with one additional parameter describing drilling stiffness. In [2] it was shown, that this approach leads to an efficient and reliable description of laminated shells behaviour that undergo large displacements. The stress state in each ply of laminated shell is calculated assuming first order shear deformation kinematics in the thickness direction.

2.2. First ply failure

Since the stress tensor of the layer’s material is in the applied shell theory not symmetric, it is not possible to apply a standard failure initiation criterion. Hence, we propose to use a modified Tsai-Wu criterion to define failure surface of laminated composites, in the following form:

\[
1 = \left( \frac{1}{X_t} - \frac{1}{X_c} \right) \sigma_1 + \left( \frac{1}{Y_t} - \frac{1}{Y_c} \right) \sigma_2 + \frac{1}{X_t X_c} \sigma_3^2 + \frac{1}{Y_t Y_c} \sigma_1 \sigma_2 + \frac{\max \left( |\tau_1|, |\tau_2| \right)}{S^2}
\]

where \( \sigma_1, \sigma_2, \tau_1, \tau_2 \) (\( \tau_1 \neq \tau_2 \)) describe stress tensor components in the material system, while \( X_t, Y_t, X_c, Y_c, S \) denote absolute values of: tensile strength in the 1st and 2nd material direction, compressive strength in the 1st and 2nd material direction and shear strength in the 1-2 plane. The difference between the failure criterion proposal given here and the original one (see for instance [3]) concerns shear contribution. Since the shear components are asymmetric now, we imply that the extreme value of shear stress will affect expression (1) in the least favourable way. This yields a more precise description of failure initiation.

3. Numerical examples

A case of compression-loaded flat panel is analysed. A similar panel (named as panel C4) is considered among others in [4] or [5], where also experimental data are given. The panel is 508mm long and 178mm wide. There are different sets of input data concerning elastic material constants, strength parameters and lamina thickness available (see for instance [4], [5]). Following [5] we use: the longitudinal elastic modulus \( E_1 = 131GPa \), the transverse elastic modulus \( E_2 = 13.031GPa \), the in-plane and transverse shear moduli \( G_{12} = G_{13} = 6.205GPa \), the transverse shear modulus \( G_{23} = 3.447GPa \), the major Poisson’s ratio \( v_{12} = 0.38 \). The strength data as explained in 2.2 are: \( X_t = 1400MPa, X_c = 1138MPa, Y_t = 80.9MPa, Y_c = 189MPa, S = 62.05MPa \). The single ply thickness is \( t = 0.132mm \). The panel

---

* Sobczyk is supported under Gdańsk University of Technology (Poland), Faculty of Civil and Environmental Engineering, Young Scientist Support Programme. The support is gratefully acknowledged. Abaqus calculations were carried out at the Academic Computer Centre in Gdańsk.
is made of 24 plies in the following configuration \( [\pm 45/90/\pm 45/90/\pm 45/90/0]_s \), where 0 direction is parallel to X and 90 is parallel to Y axis of the local element coordinate system shown in Fig. 1.

Figure 1: Geometry and boundary conditions of the panel

Calculations are performed using Abaqus 6.14 and a non-commercial FEM code CAM [1] with additional author procedures. In Abaqus S4 shell element is applied. It is 4-node fully integrated element with additional formulations preventing against locking effect. CAM calculations are based on 16-node element with full integration, in which locking effects may be regarded as negligible. In both programmes the shell behaviour is described by means of Equivalent Single Layer Approach, where each layer has 3 sectional integration points. In Abaqus a mesh is created with 24 elements along and 12 across the panel. It involves the same number of nodes as the model in [4]. On the other hand two mesh variants are studied for CAM calculations. The first one with 8 elements along and 4 across the panel determining exactly the same node distribution as in the above mentioned models. The second one comprises of 16 elements along and 8 across the studied geometry.

In both programs geometrically non-linear analysis is carried out. The panel buckles into two longitudinal half-waves [4], hence a point force imperfection (equal to 5N) is applied in the location and direction of the maximum and minimum value of out-of-plane displacements, obtained for the buckling shape by means of linear buckling analysis. Fig. 2 presents \( P_{mag} \) (edge load magnitude - see Fig. 1) versus end shortening relationship for the panel.

On the basis of Fig. 2 buckling loads can be determined, which are: \( \sim 39.1 \text{kN} \) for Abaqus and \( \sim 39.0 \text{kN} \) for CAM calculations (both mesh variants). In the course of the analysis FPF loads were determined. Failure criterion indices were checked in the Gauss points in three locations within each layer thickness (in external and middle positions). The FPF load obtained from Abaqus (original Tsai-Wu criterion [3]) is \( \sim 79.1 \text{kN} \), while CAM calculations (modified Tsai-Wu criterion) yield \( \sim 77.5 \text{kN} \) for both mesh variants. FPF loads are normalized by the buckling load and compared with the experimental value given in [4]. Consequently, they are: \( \sim 2.03 \) and \( \sim 1.99 \), respectively for Abaqus and CAM calculations. The experimental value is \( \sim 1.97 \). The failure occurs in all cases in the middle of simply supported edge in the panel outer ply. Figure 3 shows contours of modified Tsai-Wu failure criterion (1), for coarse mesh CAM calculations, outer ply (45°).

4. Conclusions

The original authors’ proposal of modified Tsai-Wu criterion under formalism of nonlinear 6 parameter shell theory [1] and particular constitutive relation [2] was compared with the original approach [3]. Numerical calculations of compression-loaded flat panel presented in this paper revealed similar panel response for the theories using asymmetric and symmetric strain tensor measures. Also first ply failure loads obtained in numerical analyses are comparable with the experimental one [4], in the field of load values and failure location. The first ply failure load value produced by the modified criterion is in good agreement with the reference one.

References

Method of prediction of load-settlement curve for a single pile

Bartłomiej Czado¹, Bogumił Wrana²
Faculty of Civil Engineering, Cracow University of Technology
Warszawska 24, 31-155 Kraków, Poland
e-mail: bczado@pk.edu.pl, bwrana@pk.edu.pl

Abstract

In this article a method of prediction of a load-settlement curve for a single pile is presented. Results of a cone penetration test (CPT) of subsoil are used for calculation of pile ultimate bearing capacity. Authors proposed the method of estimating the pile head displacement \( s \) which depends on three components: a) pile elastic deformation; b) pile head displacement resulting from deformation of soil around the pile; c) pile base displacement resulting from elastic and plastic deformations of soil under the pile base which occur when the load exceeds pile shaft capacity.

Keywords: pile foundation, cone penetration test, CPT, load-settlement curve

1. Introduction

In a traditional approach pile foundations are considered non-displacing (fixed) supports. This assumption is eligible only for piles designed with relatively high factor of safety, which results in pile displacements not exceeding 0.5-1.0% of pile diameter \( D_p \).

Nowadays geotechnical design is highly focused on reducing costs of foundation works, which leads to searching for more precise methods of predicting not only pile ultimate bearing capacity but also pile load-settlement curve. In such a case it is possible to define a non-linear supports in numerical model of a structure.

The article presents a proposition of a method of prediction of \( s(Q) \) function for a single pile based on the results of cone penetration test (CPT).

2. State of art

Two studies on this matter will be discussed in this article. First, presented by Meyer et al. [2,3], is an example of a method based on field measurements approximations (full scale pile load tests) with analytical functions. Second, proposed by Gwizdała et al. [1], is a numerical model for predicting the pile behavior before its execution, basing on information on soil profile and pile geometry.

2.1. Meyer et al. method

In this method it is assumed that the pile ultimate bearing capacity is defined as a value of load at which pile displacement increases without further increase of load. The analytical description of load-settlement relation is defined by Eq. (1).

\[
s(Q) = A \cdot \left( \frac{1}{\left(1 - \frac{Q}{Q_{\text{lim}}} \right)^{\kappa}} - 1 \right)
\]

(1)

The disadvantages of this method are that the function is highly sensitive to change of \( \kappa \) parameter (see Fig. 1) and requires determination of \( Q_{\text{lim}} \) value in a field test.

![Figure 1: Meyer, 2013 [3]](image)

The method based on data approximation from pile load tests. It does not include any information on soil profile and/or pile geometry, this means it cannot be used for pile behaviour prediction.

2.2. Gwizdala et al. method

The method developed by Gwizdala and his team is based on discrete numerical model of a pile. The assumptions of the model and an example of results are shown in Fig. 2.

The advantage of this method is that it allows prediction of pile load-settlement curve. On the other hand there is no analytical formula available for use in modelling of a structure.

![Figure 2: Gwizdala and Dyka, 2001 [1]; Stęczniewski, 2003 [4]](image)
3. Proposed method

A method that combines the advantages of both approaches presented above is discussed in the paper. At the current stage of research it was only verified on one site (results of 5 static load tests), so it is considered as an early-stage proposition.

3.1. Main assumption

It is assumed that non-linear function describing load-settlement curve of a pile reaches a value of \( s_{ult} = 0,1D_p \) at a load value of \( Q_{ult} \) – ultimate bearing capacity of the pile (see Fig. 3).

![Assumed definition of pile ultimate capacity](image)

**Figure 3: Assumed definition of pile ultimate capacity**

3.2. Model description

Pile head displacement \( s \) (settlement) depends on three components:

a. pile elastic deformation \( s_{el} \) which depends on external load and distribution of soil friction resistances along the pile shaft,

b. pile head displacement \( s_s \) resulting from deformation of soil around the pile,

c. pile base displacement \( s_p \) resulting from elastic and plastic deformations of soil around the pile base which occur when the load exceeds pile shaft capacity.

Determination of the first component requires description of compressive force distribution in a pile. A function used for prediction was developed basing on the results of two pile load tests performed on instrumented piles [5]. The resulting function of pile elastic deformation is shown in Fig. 4.

Function used for descriptions of second and third component are defined by Eqs (2) and (3) respectively

\[
s_s(Q) = 0.5 \cdot s_{ult} \cdot e^{\frac{1}{b} \ln \left( \frac{Q}{Q_{ult}} \right)}
\]

(2)

\[
s_p(Q) = 0.5 \cdot s_{ult} \cdot e^{\frac{1}{b} \ln \left( \frac{F_b(Q)}{Q_{ult}} \right)}
\]

(3)

where: \( s_{ult} = 0,1 \cdot D_p \); \( Q_{ult} \) is a value of pile ultimate bearing capacity based on calculations with a use of CPT results; \( F_b(Q) \) is a function defined as shown in Fig. 5.

![Assumed function of pile elastic deformation \( s_{el} \)](image)

**Figure 4: Assumed function of pile elastic deformation \( s_{el} \)**

4. Results

The model was verified on the site of Centrum Jana Pawła II in Kraków-Łagiewniki. Soil profile used for calculations was taken from seven CPT tests performed on the same site. The predicted curve (solid line) with its three components (dashed lines) vs. the results of pile load tests (dotted lines) are shown in Fig. 6.

![Prediction compared to tests results](image)

**Figure 6: Prediction compared to tests results**

5. Conclusions

In the article the authors suggested a new way to estimate load-settlement curve of a single pile foundation based on CPT test results. At present, the proposed method is checked on currently executed piles.

References


Design of a low power wind turbine adjusted to near-ground higher turbulence

Rafał Tadeusz Dalewski1, Robert Jóźwiak2, Olgierd Kobyliński3, Krzysztof Rafal4, Jacek Szumbarski5

1,2,5 Faculty of Aeronautical and Power Engineering, Warsaw University of Technology
Nowowiejska 24, 00-665 Warszawa, Poland
e-mail: rdalewski@meil.pw.edu.pl

3 BNC Sp. z o.o.
Świeradowska 44, 02-662 Warszawa, Poland

4 Faculty of Aeronautical and Power Engineering, Warsaw University of Technology
Nowowiejska 24, 00-665 Warszawa, Poland

Abstract

The aim of the presented process was to develop a fixed stroke, low-power (5 and 10 kW) wind turbine adjusted to Polish environmental conditions. During the process a family of new aerodynamic airfoils adjusted to low-speed, high turbulent conditions was designed, further utilized in fixed stroke turbine blades. Design was obtained by means of numerical, analytical and experimental methods.

Keywords: wind power, wind turbines, low Reynolds, high turbulence, CFD, experimental methods

1. Introduction

This is to present a multistage design and optimization process of a fixed-stroke low-power wind turbines adjusted to Polish environmental conditions. A need of creating such a turbine was triggered by a “prosumental” power harvesting policy oriented at distributed (home-installed) power generators of polish government. One of such a method of energy harvesting is usage of wind power generators.

Although we can observe hundreds of turbine designs most of them is not very suitable for Polish environment. For Poland the wind speed is rather low (mean 3-5 m/s), and because of general conditions for installation, turbines should be adjusted to near ground, high turbulent, low-speed conditions. Therefore the presented design was aimed at early start solution, adjusted to higher level of turbulence, efficient for low speeds (3-5 m/s) and rated for lower velocity i.e. 10 m/s.

2. Early stage design

Generally design conditions formed above are not particularly beneficial for aerodynamic devices. Low speed and small dimension constitute low Reynolds number regime, increasing drag components and laminar separation risk. Because of that the first stage was to identify most probably-occurring conditions. On the basis of common knowledge and local wind studies it was decided that a family of profiles will be developed for Reynolds number close to 300,000 and free stream turbulence app. 0,25%.

Profiles were designed using Xfoil code [2], and further experimentally investigated using low-turbulence tunnel for various turbulence values.

Profiles characteristics are presented below (Figs. 1-3).
Further profiles were tested taking into account sensitivity for shape variation and various turbulence levels. Finally a set consisting of 10%, 12%, 15%, 18% and 24% profile was developed.

3. Blade design

Blades were designed with usage of hybrid BEM method with Prandtl correction for hub and tip [1]. Two further assumptions influenced the design – fixed stroke and low-losses generator with mechanical breaks and control system allowing for occasional over loading and rotational speed setting. The main optimization factor was then a compromise between maximum efficiency and relatively high moment value, especially for low speed regime. Low generator starting moment allowed for finding a good balance between moment produced and efficiency. This was found for a constant TSR 4.3 for 5 kW turbine and 4.4 for 10kW, although control low for the turbine assumes TSR adjustment for various wind velocities. Two blades were proposed – one for 5kW turbine (rated at stream velocity 10 m/s), and for 10 kW. Their performance was shown in figures 4-6, where 4 presents a power curve, 5 – aerodynamic efficiency and 6 – moment and thrust produced by 5kW turbine.

Finally a model (1m diameter) was tested in new environmental installation of Warsaw University of Technology (Politechnika Warszawska) and analysed using commercial CFD packages. This helped for blade optimization and a tip design. Final results of design process taking into account numerical, analytical and experimental analysis of turbines and their model will help in understanding of low speed aerodynamics and provide modern wind power generator design for Polish and Central-European environment.

References

Influence of the weld geometry on the Stress Intensity Factor (SIF) of the cylindrical welded joint subjected to complex load

Jelena M. Djoković1*, Ružica R. Nikolić2*, Jan Bujnak3*

1Technical Faculty in Bor, University of Belgrade
Vojske Jugoslavije 12, 19210 Bor, Serbia
e-mail: jelenamdjokovic@gmail.com
2Faculty of Engineering, University in Kragujevac
Sestre Janjić 6, 34000 Kragujevac, Serbia and Research Center, University of Žilina
Univerzitna 1, 010 26 Žilina, Slovakia
e-mail: ruzicarnikolic@yahoo.com
3Faculty of Civil Engineering, University of Žilina
Univerzitna 1, 010 26 Žilina, Slovakia
e-mail: Jan.Bujnak@fstav.uniza.sk

Abstract

Cylindrical welded joint, subjected to axial tension, bending and torsion is considered in the paper for determining the influence of the weld geometry of the fracture mechanics parameters. Stress intensity factors are calculated analytically, based on the Linear Elastic Fracture Mechanics (LEFM) concept, taking into account the weld and load through the corresponding correction factors. The obtained results show that the normalized Mode I stress intensity factor (SIF) decreases with increase of the crack length, in the case of the triangular fillet weld, while the normalized Mode III stress intensity factor increases. Identical conditions occur for the case of the convex fillet weld. For the case of the concave fillet weld, both stress intensity factors are increasing with the crack length increase. Based on the presented results, one can conclude that, from the aspect of the fracture resistance, the convex fillet weld exhibits the best properties.

Keywords: cylindrical welded joint, corner joint, stress intensity factor

1. Introduction

Welding is a widely used technique for connecting the structural elements of various sizes and geometry and for transfer of loads from one element of a structure to another. The welded joint resistance is of the utmost importance for integrity of such a structure.

When the parts are joined by fillet welds, a surface of geometrical discontinuity appears. When two structural parts are leaned against each other, the filler metal creates a fillet weld. Here the face surface of the thinner part is not connected to the surface of the thicker part, Figure 1, creating a geometrical discontinuity. It behaves like a crack, whose size is equal to thickness of the thinner material. Analysis of the fillet weld geometry is very important for obtaining the higher fracture resistance of these welds.

The resistance of welded joints, from the fracture mechanics aspect, has been a subject of interest of many researchers [1-5]. Influence of the fillet weld shape and its geometry on the fracture resistance of cylindrical structural elements is analyzed in the paper. Determination is done of the SIFs with the application of the Linear Elastic Fracture Mechanics (LEFM) concept. The considered problem is presented in Fig. 1. The axle of radius \( r \) is welded to a disk of radius \( R \) and length \( L \). The welded cylindrical part is loaded by the axial tensile force \( F \), bending moment \( M \) and the torque \( T \). As can be seen from Fig. 1, there is an unwelded area between the two parts of diameter \( 2a \), which is considered a crack. The weld dimensions are height \( h \) and width \( w \). It could be triangular or rounded with radius \( \rho \) (convex or concave).

2. Determination of the stress intensity factor

The stress and displacement fields at the crack tip are characterized by the SIFs, \( K_I, K_{II} \) and \( K_{III} \), which are in LEFM defined as

\[
K_i = \lim_{r \to 0} \sqrt{2\pi r} \sigma_i, \quad K_{II} = \lim_{r \to 0} \sqrt{2\pi r} \sigma_{II}, \quad K_{III} = \lim_{r \to 0} \sqrt{2\pi r} \sigma_{III}, \quad (1)
\]

for \( \theta = 0 \), where \( r \) and \( \theta \) are the polar coordinates with the coordinate frame origin at the crack tip.

The SIF depends on the shape of the part and on the loading conditions. To calculate the SIF, one needs to determine the complete stress field at the crack tip and to calculate the limit value from Eqn (1). For the practical application, it takes too much time, so usually other approaches are used.

For the welded joints, the SIFs obtained due to Ref. [6]:

\[
K_i = Y_i \cdot C_i \cdot \sigma_i \sqrt{2a}, \quad (2)
\]

where \( \sigma_0 \) is the reference load, \( Y_i \) is the dimensionless parameter that depends on the sample geometry and applied load and \( C_i \) is

* This research was partially supported by the Ministry of Education, Science and Technological Development of Republic of Serbia through Grants ON174001, ON174004 and TR32036 and by European regional development fund and Slovak state budget by the project "Research Center of the University of Žilina " - ITMS 2622020183. The authors are very grateful for this funding.
the correction factor that accounts for the stress concentration due to the presence of the weld.

For the problem shown in Fig. 1, axial tension and bending dominantly influence the Mode I SIF $K_I$, while their influence on Mode II SIF $K_{II}$ is negligible thus not considered within this analysis. Torsion influences Mode III SIF.

The Mode I SIF $K_I$ based on Eqn (1) can be written as:

$$K_I = Y_F C_F F + Y_M C_M 4Ma \left( r + a^2 \right) \left( r - a^2 \right) \sqrt{\pi a} ,$$  \hspace{1cm} (3)

where $Y_F$ and $C_F$ are the dimensionless parameter and the load correction factor for the axial force, respectively while $Y_M$ and $C_M$ are the same for the bending moment, see Ref. [7].

The Mode III SIF $K_{III}$, can be written as:

$$K_{III} = Y_T C_T 2Ta \sqrt{\left( r - a \right)} \sqrt{\pi a} ,$$  \hspace{1cm} (4)

where $Y_T$ and $C_T$ are the dimensionless parameter and the load correction factor for the torque, respectively, [7].

3. Results and discussion

Figures 2 and 3 present normalized Mode I and Mode III SIFs variation in terms of normalized crack length for three different fillet weld geometries determined by analytical expressions (3) and (8), respectively and by application of the symbolic programme Mathematica®. The normalization factor for SIFs is $1 \text{[MPa]} \cdot \sqrt{\pi \cdot 0.01 \text{[m]}}$. The dimensions and loads are $s/r = w/r = 1$, $F = 1 \text{kN}$, $M = 1 \text{kNm}$ and $T = 1 \text{kNm}$. Material properties are $E = 210 \text{GPa}$ and $\nu = 0.3$.

Figure 2 shows that normalized Mode I SIF is almost constant, with small decrease with the crack length increase for the triangular and convex fillet welds, while for the concave fillet weld it increases with crack length. The normalized Mode III SIF increases with crack length increase for all the three fillet weld shapes (Fig. 3).

4. Conclusion

In the paper the welded joint of two cylindrical parts is analyzed, with different shapes of the fillet weld, triangular, convex and concave. The welded joint is subjected to axial tensile force, bending and torsional loading.

From the aspect of fracture resistance it may be stated that the convex fillet weld exhibits the best properties.

References


Detection of damages in a riveted plate
Łukasz Doliński1, Marek Krawczuk2, Magdalena Palacz3, Arkadiusz Żak4
1,2,3,4Faculty of Electrical and Control Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: mpalacz@pg.gda.pl

Abstract

The paper presents the results of damage detection in a riveted aluminium plate. The detection method was based on a Lamb wave propagation. The plate was analysed numerically and experimentally. Numerical calculations were carried out by the use of the time-domain spectral finite element method, while for the experimental analysis the laser scanning Doppler vibrometry (LSDV) was applied. The panel was excited by a 5-pulse sinusoidal wave packet of a 35 kHz carrier frequency. A comparison of the experimental and analytical result indicates a good quality of the proposed damage detection technique.

Keywords: wave propagation, damage detection, numerical model

1. Introduction

Elastic waves are a type of mechanical waves propagating in an elastic medium due to of forces associated with its volume deformation (compression and extension) as well as shape deformation (shear) [1]. One of the type of elastic waves were Lamb waves, that propagate in infinite media bounded by two surfaces and arise as a result of superposition of multiple reflections of longitudinal waves and shear waves from the bounding surfaces. In the case of these waves medium particle oscillations are very complex in character. Various anomalies in wave propagation patterns resulting from wave-damage interaction can be observed, interpreted and employed for damage assessment. It is well-known that the presence of damage results in reflection and scattering of propagating elastic waves and such features are commonly used for damage detection purposes. Wave propagation may be analysed in two ways – experimentally and numerically. For a practical wave analysis laser vibrometry may be depicted as one of the most efficient tools.

In the presented example of damage detection a comparison of two approaches, numerical and experimental, was done.

2. Numerical model

For numerical modelling the time-domain spectral finite element method [2] is recommended due to its simplicity of application. The method originates from the application of spectral series for the solution of partial differential equations, while at the same time its basic ideas are analogous to the classical finite element [3] approach. Its main assumption is the application of orthogonal Lobatto polynomials as approximation functions defined at appropriate Gauss-Lobatto-Legendre integration points. As a consequence the inertia matrix obtained in this spectral approach is diagonal making the total cost of numerical calculations much less demanding. Additionally, thanks to the orthogonality of the approximation polynomials the spectral finite element method is characterised by exponential convergence [1].

In the current formulation of a spectral shell element according to the time-domain SFEM as approximation functions Lobatto polynomials of the first kind were used. As an example the 5th order polynomials were chosen. The resulting grid of element nodes in this case is presented in Fig. 1.

Figure 1: Displacement components in a 36-node spectral shell finite element

In the current formulation of the isotropic shell element the Lobatto node distribution was used based on the 6-th order complete Lobatto polynomial [1]. In the normalised (curvilinear) coordinate system \( \xi, \eta, \zeta \) of the element the coordinates of the element nodes \( \xi (i=1,\ldots,6) \) and \( \eta (j=1,\ldots,6) \) were assumed as the roots of:

\[
L_5^i (\xi) = 0, \quad i = 1,\ldots, 6
\]

\[
L_6^j (\eta) = 0, \quad j = 1,\ldots, 6
\]

where the complete Lobatto polynomial is defined in the following manner [25]:

\[
L_0^i (x) = \left( 1 - x^2 \right)^L_0 (x) = \left( 1 - x^2 \right)^P_0 (x).
\]

where \( L \) is the 4-th order Lobatto polynomial, \( P \) is the 5-th order Legendre polynomial and the symbol ‘ denotes differentiation in respect of \( x \).

In the present definition of the spectral shell finite element an extended form of the displacement field is employed [4]. The element has six degrees of freedom per node including two in-plane displacement components \( u, v \) and four out-of-plane displacement components \( w, \Phi, \Psi, \Omega \) as well as \( \Omega \), as shown in Fig. 1. The reader is kindly encouraged to see [4] or [1] for a similar solution technique used for the element definition.
3. Case description

For numerical and experimental test prepared an aluminium panel (E=72.7 GPa, ν=0.33, ρ=2700 kg/m³) was with two T-shaped stiffeners. Dimensions of the plate were: 1x0.7x0.001 m and of the stiffeners: 0.04x0.04x0.004 m. The stiffeners were fixed with 54 rivets located in 2 parallel rows. The assumed additional mass has not been bigger than 1% of the total riveted plate.

For numerical modelling of the described element there has been used 6 node spectral element based on 3-mode theory of higher order, shortly given in previous paragraph. Damages were modelled by modulation the magnitudes of mass and stiffness in adequate elements. The model built for numerical simulations consisted of 54761 nodes in total. The calculation time was 0,5 ms for the analysis of 8000 time steps.

In both cases damage was tested by some removed rivets as well as additional masses attached to the plate surface. For laser measurements the specimen has been isolated from the influence of external vibration source by locating it on polyurethane foam. For excitation, a PZT actuator (T216-A4NO-273X) has been used with resonant frequency of 7,3 kHz. All data were measured using a Polytec PSV-400 SLDV. As Lamb wave source was used a sinusoidal signal of 35 kHz frequency multiplied by Hanning window, covering 5 periods of a sinusoid. All signals were generated by a digital generator of ±10V output. The signals were then amplified to ±100V.

During measurements were taken velocities at every measurement point in the direction perpendicular to the plate surface. The signal has been quantified to 1024 time samples and every measurement was repeated 10 times and the average value was calculated in order to increase of the signal–noise ratio. The time between every excitation was long enough to dump previously activated wave. Also the release time was determined for every periodic frequency by registering in a sample point the time necessary to attenuate the wave totally.

Figure 2 shows a comparison of results obtained for excitation of 35 kHz. For the measured signals (right hand side) the value of Root Mean Square has been recalculated for every measurement point.

4. Conclusions

The paper presents a comparison of the results obtained during numerical calculations and experimental measurements. It was shown that the proposed numerical model based on higher mode theory is suitable to model a real riveted panel for the analysis of such a complicated phenomenon, like elastic wave propagation. From practical point of view it is worth to emphasize that numerical simulation of complicated processes may allow significant cost saving of different prototypes. This is why the development of efficient numerical tools is still important for the mechanical applications.

It was also shown that using wave propagation for damage detection makes it possible to identify isolated damaged rivets. On the basis of performed research results it may be concluded that adequate selection of analysis parameters for wave-based damage detection techniques guarantees a very sensitive damage diagnostics tool.

References


Figure 2: Comparision of numerical (a) and experimental (b) results
Application of experimental modal analysis and wavelet transformation for damage localisation in a composite wind turbine blade

Łukasz Doliński¹, Marek Krawczuk²

¹,² Faculty of Electrical and Control Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: lukasz.dolinski@pg.gda.pl¹, marek.krawczuk@pg.gda.pl²

Abstract

The article is focused on the problem of non-destructive damage localisation in the outer composite skin of a wind turbine blade applying utilisation of the Laser Doppler Scanning Vibrometry (LDSV). The proposed method is based on wavelet analysis of modal parameters, which are determined experimentally. The basic assumption is that mode shapes of free vibrations depend on the physical properties of the structure investigated. Experimental investigations were carried out by the authors on a specially manufactured composite blade. Damage was simulated by additional high stiffness elements fixed onto the surface of the blade. The measurements were performed by the use of the LDSV measurement technique. The main advantage of the LDSV is its high sensitivity and non-contact measurement capability. Series of measurements were carried out for several localisations of damage and next bending mode shapes of the blade were analysed by a one-dimensional continuous wavelet transform. The analysis of all damage cases shown that the identification and localisation of a relatively small damage is possible.

Keywords: modal testing, wavelet transformation, non-destructive damage detection, wind turbine blade, vibration based methods

1. Introduction

Wind turbine blades are the most important parts of a wind power station. The continuous character of their work causes that the blades are unceasingly loaded in different directions, what involves special requirements concerning their strength and reliability. Because of that methods for damage detection in the outer composite skin of the blades is investigated by many researchers [1–3]. The majority of classical diagnostic techniques such as ultrasonography or acoustic emission requires a full-stop of the rotor on the time of tests. Investigation results obtained from such tests determine if any damage changed the structure vibration spectrum, which, on the other hand, allows to create non-destructive damage detection methods for the blades during their normal working conditions. Damage may significantly reduce structural stiffness, which leads to changes in natural frequencies and mode shapes. The main goal of the investigation is to develop a non-destructive damage detection method for damage detection and localisation in a composite outer skin of wind turbine blades using vibration parameters (mode shapes of free vibrations), which depend on the structure physical properties.

2. Experimental setup

The geometry of a composite wind turbine blade under investigation is presented in Fig. 1. The length of the blade is 1.74 meter. Glass fibres and epoxy resin were used as laminate components. The reinforced fibres were arranged symmetrically [±45°]. The blade was divided into 3 sections, each of them of a different number of material layers. During the measurements the blade was placed on a special table, which isolated the blade from external vibration sources and which was connected to an electromechanical shaker used for vibration excitation. The experiment was carried out with the use of a PSV- 400 Laser Doppler Scanning Vibrometer unit by Polytec, Ltd., which enabled fast, accurate and non-contact vibration measurements.

The unit was equipped with a precise optical transducer used for the determination of the speed of vibrating measurement points. The transducer measured changes in the frequency of the light reflected from vibrating surfaces.

Figure 1: A composite blade under investigation

3. Theoretical background

Mode shapes of natural vibrations obtained experimentally were analysed by a one-dimensional continuous wavelet transformation (CWT). The results of simulation studies presented in [4] show that the method proposed is useful in damage detection. The wavelet transformation is a kind of mathematical function used to divide a given continuous-time signal into different frequency components and to study each component with a resolution that matches its scale. Wavelets are scaled and translated copies of a finite-length or fast-decaying oscillating waveform ("mother wavelet"). Wavelets are well-suited for representing functions that have discontinuities or sharp peaks, and for precise deconstructing and reconstructing of finite, non-periodic and/or non-stationary
signals. Results of CWT are coefficients that determine similarity between selected waves and the signal analysed [5,6].

4. Results

The method proposed by the authors is based on a wavelet transform analysis of modal parameters. The basic assumption of damage detection based on vibration measurements is that characteristic parameters, like natural frequencies or mode shapes of free vibrations depend on physical properties of the structure. The first ten mode shapes of the undamaged and damaged blade under consideration were determined by velocity measurements along the blade (one line). Only the bending form was taken into account in the analysis. An example of measurement results is presented in Fig. 2. The mode shapes of the blade were analysed by a one-dimensional continuous wavelet transform, while damage to the blade was simulated by additional high stiffness elements fixed onto its surface. Series of measurements were carried out for 3 damage localisations and 3 damage lengths. Necessary calculations were carried out for different kinds of wavelets in order to find the optimal solution. The best results for boundaries localisation were obtained using Gaussian wavelet of the 4th order. Exemplary results for damaged specimen (8th mode, damage in the middle of the measured length) obtained by the authors are presented in Fig. 3.

A solution to this problem was the application of a reference signal obtained from measurements on an undamaged blade. Wavelet coefficients for a differential signal between the damaged and undamaged state are presented in Fig. 4.

Figure 2: Example of measurements results – 8th eight

Figure 3: Wavelet coefficients for 8th mode – vertical lines indicate damage boundaries

Figure 4: Wavelet coefficients for 8th mode – vertical lines indicate damage boundaries

Figure 5: Wavelet coefficients for a differential signal between the damaged and undamaged state

Successive tests aimed to determine the minimal number of measurement points, which would allow to detect damage in the blade under consideration.

5. Conclusions

A method for damage localisation in the outer skin of a composite wind turbine blade, based on one-dimensional continuous wavelet analysis applied to mode shapes, was presented. Mode shapes of natural vibrations required for the analysis were determined experimentally. A Gaussian wavelet with different decomposition levels was applied as a mother wavelet. The analysis of all damage cases shown that the damage detection based on wavelet techniques can effectively identify and localise damage in wind turbine blades.

References


Investigation of turbulent boundary layers at moderate Reynolds number in the vicinity of separation

Artur Dróżdż¹, Witold Elsner², Adam Kępiński³

¹,²,³ The Faculty of Mechanical Engineering and Computer Science, Częstochowa University of Technology
Dąbrowskiego 69, 42-201 Częstochowa, Poland
e-mail: arturdr@imc.pcz.czest.pl, welsner@imc.pcz.czest.pl

Abstract

The paper deals with the experimental analysis of turbulent boundary layer at the flat plate developed under the influence of strong adverse pressure gradient. The special design of the test section equipped with perforated, movable upper wall allows to generate on the bottom flat plate the turbulent boundary layer, which is at the verge of separation. The emphasis is on the analysis of the streamwise fluctuating velocity and mean velocity profiles as well as on some criterion, capable to predict the detachment position. The change of mean velocity profiles and fluctuating velocity fields confirmed the presence of specific conditions for the separation of the turbulent layer. The increase of APG strength cause the decrease in mean velocity below the log law. The effect of APG is especially visible in significant modification of the distribution of Reynolds stresses.

Keywords: turbulent boundary layer, adverse pressure gradient, separation

1. Introduction

In a number of types of near wall flows the turbulent boundary layers (TBLs) subjected to an adverse pressure gradient (APG) are in the spotlight. That is because TBLs under the influence of the adverse pressure gradient (APG) are frequently encountered in many engineering applications, such as diffusers, compressor and turbine blades, and the trailing edges of airfoils. The performance of such flow devices is significantly affected by the presence of the adverse pressure gradient. It is due to the inner part of the boundary layer nearest to the wall, which is crucial in determining the skin friction drag. If a turbulent boundary layer flow encounters a strong APG, the flow becomes unstable, if the APG is sufficiently large, it separates from the surface. The existence of separation involves an increase of energy losses connected sometimes with pressure and velocity fluctuations. The evidence of the latter phenomenon was given by Cherry et al. [1], who investigated the unsteady structure of a separated and reattaching flow. Unstable location of turbulent separation results also from the impact of vortex structures that fall into the area of separation, causing a temporary increase in momentum. An extensive phenomenological description of the flow separation distinguishing various stages of separation was made by Simpson [2]. The turbulent boundary layer that is maintained on the verge of separation has already been studied experimentally [3]. The authors showed that the flow close to separation exhibited a definite non-equilibrium character, indicated by the different scales required for collapse of the mean velocity and turbulence intensity profiles.

The pressure gradient and especially the APG have a complex effect on the mean velocity profile and so on the shape factor $H \equiv \delta'/\theta$, which was shown to increase in the presence of an adverse pressure gradient [3]. The effect of APGs is also visible in significant modification of the distribution of Reynolds stresses. This effect was reported by many authors for different pressure gradients, as for example by Krogstad and Skare [3] and Nagano and Houra [4]. The similar observations were published by Materny et al. [5], who demonstrated the appearance, independently of the first peek in the inner layer, of the second maximum in profiles of velocity fluctuations. The second maximum moves away from the wall with pressure gradient but the underlying mechanism of this effect is still not understood well. It is believed that it is associated with the production and breakdown of organized flow structures in the same sense as it is proposed for the transition process to turbulence in wall-bounded flows and based on streak instabilities.

The experiments were performed in an open-circuit wind tunnel, where the turbulent boundary layer developed along the flat plate, which is 7 m long. A newly developed test section located at the end of the wind-tunnel is equipped with perforated, movable upper wall enabling to generate flow conditions for the separation of turbulent boundary layer at the bottom flat plate. The profile of the upper wall was adjusted initially to have a constant pressure gradient along the test section. In order to accelerate the transition to turbulent boundary layer the tripping wire followed by the strip of coarse-grained sandpaper was used. Finally, it was allowed to obtain, for inlet velocity equal $U=15m/s$, the value of Reynolds number, based on the momentum loss thickness $\theta$, equal $Re_\theta \approx 8300$.

The measurements were performed with a hot-wire anemometry CCC developed by Polish Academy of Science in Krakow. A single hot-wire probe of a diameter $d = 3 \mu m$ and length $l = 0.4 \ mm$ was used. Acquisition was maintained at frequency 50 kHz with 30 s sampling records.

2. Results

The measurements were performed for two pressure gradient conditions controlled by means of upper wall position and by flow suction. The first conditions were set in order to achieve turbulent boundary layer on the verge of separation at the end of test section and the second one with higher pressure.
gradient to shift the separation point upstream. The flow conditions are characterised by pressure gradient parameter \(C_p=1-(U_e/U_{e0})^2\) (see Fig.1). To identify the separation point the criterion proposed by Sandborn and Kline \([6]\) has been used. They showed that the shape factor defined as:

\[
H_{sep} = 1 + \frac{1}{1 - \frac{\delta'}{\delta}}
\]

has a value 2.7 at the Intermittent Transitory Detachment (ITD) position. The Intermittent Transitory Detachment is when the reverse flow occurs of about 20% of the time \([2]\).

Figure 1 shows \(C_p\) and shape factor \(H\) for analysed flows. It is seen that, between \(x = 200\text{–}1000\) mm, for the \(s\) (strong) case the \(C_p\) is more inclined. These changed conditions forced the earlier boundary layer separation, what is confirmed by the significant change of \(H\) distribution. It is due to the fact that the displacement thickness at separation increases sharply. Using the criterion introduced above, it can be concluded that for this case, at \(x = 1000\) mm \(H_{sep}\) is equal 2.74, which means that the ITD point is reached. On the other hand for the \(w\) (weak) case, at \(x = 1400\) mm \(H_{sep}\) is equal 2.42.

In Figure 2 and 3 a strong deformation of velocity profile and the disappearance of near-wall maximum of streamwise Reynolds stresses are seen. The stronger decrease is however, observed for the second case, where near the wall the extremely low values retain until \(\gamma/\delta = 0.1\). It confirms that the turbulent boundary layer is on the verge of separation for both pressure gradient conditions, but clearly for the different locations.

References


Nonlinear model of spacecraft relative motion in an elliptical orbit

Piotr Felisiak 1, Krzysztof Sibilski 2, Wiesław Wróblewski 3

1, 2, 3 Faculty of Mechanical and Power Engineering, Wrocław University of Technology
Wybrzeże Wyspiańskiego 27, 50-370 Wrocław, Poland

Abstract

Modern control strategies, such as model predictive control, require accurate models of the process in time domain. This paper presents a formulation of nonlinear, time-variant model of a spacecraft motion relative to body orbiting in an elliptical orbit. The model assumes variable mass of the spacecraft, since the mass of expelled propellant acts strongly on system dynamics. The model is described by the state-space representation, while the parameters of the model are dependent on the time, state and control signal only. The presented model can be applied for control of spacecraft relative motion, e.g. control of a spacecraft during orbital rendezvous, one of the key space technologies. Formulation of the model is suitable for state observers, especially of some of the states unmeasured, when the model-based estimation is as accurate as possible.

Keywords: spacecraft relative motion, orbital rendezvous, orbital mechanics, astrodynamics, nonlinear model

1. Introduction

The reason of investigation presented in the paper was the need for a model formulation suitable for a model predictive control algorithm. One of the design guidelines was the ability to control the spacecraft in a case while the reference point moves in an elliptical orbit. Typically, the models of a spacecraft relative motion result from linearization of full nonlinear equations. In the case of circular orbit problem, the example is the well-known Hill-Clohessy-Wiltshire model. Several models were derived for a relative motion in an elliptical orbit, such as Tschauner-Hempel equations or Yamanaka-Ankersen equations. However, all of these linearized models are valid only for small separations between the spacecraft and the reference point. The model presented in this paper is an augmentation of a full, nonlinear model.

One of the requirements to the model was the description in time domain, while this kind of models are usually formulated in domain of the so-called true anomaly, the angular position in the orbit. Due to a problem numerical solution of the Kepler’s equation was done. Since it is assumed that the spacecraft propulsion operates by means of expulsion of a significant amount of mass, the principal model of relative motion was augmented using a mass model.

The spacecraft relative motion presented in this paper describes the motion of the so-called deputy satellite, frequently referred to as chaser satellite, relative to the chief satellite, often called a target satellite. It is assumed that the deputy satellite is controllable, while the chief satellite is uncontrolled, moving along a known elliptical orbit. The chief satellite determines a reference point. The model presented in this paper is an augmentation of a full, nonlinear model.

2. Reference frame

In order to describe the deputy spacecraft motion relative to the chief satellite, a Cartesian local-vertical local-horizontal (LVLH) coordinate frame is used. This reference frame is sometimes referred as Hill frame. The frame is attached to the chief satellite and rotates with the chief radius vector \( \mathbf{r}_c \), as shown in Figure 1.

Figure 1: Local-vertical local-horizontal (LVLH) coordinate frame

The orientation of LVLH frame is determined by the unit vector triad \( \{ \hat{o}_r, \hat{o}_\theta, \hat{o}_h \} \) where vector \( \hat{o}_r \) lies in the chief’s radial direction, \( \hat{o}_h \) is parallel to the orbit angular momentum vector, and \( \hat{o}_\theta \) completes the right-handed orthogonal triad. Then, the position of the deputy relative to the chief satellite can be expressed by a Cartesian coordinate vector \( \rho \):

\[
\rho = x_1 \hat{o}_r + x_2 \hat{o}_\theta + x_3 \hat{o}_h
\]  

(1)

3. Principal model

Derivation of the exact nonlinear equations of relative motion in LVLH frame, further called NERM, can be found in Ref. [1]. The impact of the external control force has been included here the equations are transformed into a state-space representation.
The state vector is defined:
\[ \mathbf{x}_{rm} = [x_1 \ x_2 \ x_3 \ x_4 \ x_5 \ x_6]^T \]  
(2)

where \( x_1, x_2 \) and \( x_3 \) are components of the deputy satellite relative position vector in LVLH frame, according to Figure 1. Let \( x_4, x_5 \) and \( x_6 \) be components of the relative velocity vector. Furthermore, a following control force vector is assumed:
\[ \mathbf{u} = [u_1 \ u_2 \ u_3]^T \]  
(3)

where the components \( u_1, u_2 \) and \( u_3 \) are in the radial, in-track and cross-track directions, respectively. Then, the state-space representation of the principal model of relative motion takes the form:
\[ \mathbf{x}_{rm} = \mathbf{A}_{rm} \mathbf{x}_{rm} + \mathbf{B}_{rm} \mathbf{u} + \mathbf{V}_{rm} \]  
(4)

wherein the state matrix is defined as:
\[ \mathbf{A}_{rm} = \begin{bmatrix} 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ \frac{j^2 - \mu}{r_d^3} & -\frac{2j}{r_d^2} & 0 & 0 & 2 \dot{j} & 0 \\ \frac{2j}{r_d^2} & \frac{j^2 - \mu}{r_d^3} & 0 & -2 \dot{j} & 0 & 0 \\ 0 & 0 & -\frac{\mu}{r_d^3} & 0 & 0 & 0 \end{bmatrix} \]  
(5)

where \( f \) denotes true anomaly (angular position in the orbit) of the chief satellite, \( \mu \) is the standard gravitational parameter, \( r_c \) is the current radius of the chief satellite orbit and \( r_d \) is the current radius of the deputy satellite orbit.

The input matrix is:
\[ \mathbf{B}_{rm} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ \frac{1}{m_d^2} & 0 \\ 0 & \frac{1}{m_d^2} \\ 0 & 0 \end{bmatrix} \]  
(6)

where \( m_d \) is the current mass of the deputy satellite.

Finally, the matrix of the nonlinear term can take:
\[ \mathbf{V}_{rm} = \begin{bmatrix} 0 & 0 & \mu \left( \frac{1}{r_c^2} - \frac{\mu}{r_d^2} \right) & 0 & 0 \end{bmatrix}^T \]  
(7)

4. Mass model

The mass model describes the flow rate of a propellant mass, next, the current mass of the deputy satellite \( m_d \) is calculated. The general concept of the model is based on the relationship between mass flow rate and the thruster parameter known as the specific impulse \([2]\). The relationship is given below:
\[ \dot{m} = \frac{F_{thrust}}{I_{sp} g} \]  
(8)

where \( \dot{m} \) denotes the mass flow rate, \( F_{thrust} \) is the force obtained from the thruster, \( g \) is the acceleration at the Earth’s surface and \( I_{sp} \) denotes the specific impulse. Assume that \( \mathbf{x}_{ep} \) of dimensions \( \text{dim}[\mathbf{x}_{ep}] = 1 \times 1 \) denotes mass of the propellant expelled by the deputy satellite thrusters. Next, the state-space model of the expelled mass can be expressed as:
\[ \mathbf{x}_{ep} = \mathbf{A}_{ep} \mathbf{x}_{ep} + \mathbf{B}_{ep} \mathbf{u} \]  
(9)

Since the expelled mass \( \mathbf{x}_{ep} \) does not affect the mass flow rate, the state matrix \( \mathbf{A}_{ep} = 0_{1,1} \).

The input matrix \( \mathbf{B}_{ep} \) of dimensions \( \text{dim}[\mathbf{B}_{ep}] = 1 \times 3 \) contains components corresponding to mass flow rate in the radial, in-track and cross-track directions, respectively. The components are formulated based on Equation 8, however the sign of each component depends on the sign of corresponding component in the control vector \( \mathbf{u} \). In the case where the expelled mass \( \mathbf{x}_{ep} = \|\mathbf{x}_{ep}\| \) is less than the initial propellant mass \( m_{ip} \) available to the deputy satellite, \( i \)-th component \((i = 1,2,3)\) of the \( \mathbf{B}_{ep} \) row matrix is defined as:
\[ \forall \mathbf{x}_{ep} < m_{ip} \{ u_i \geq 0 \Rightarrow \mathbf{B}_{ep1,i} = \frac{1}{I_{sp} g} \} \]  
(10)

\[ \forall \mathbf{x}_{ep} < m_{ip} \{ u_i < 0 \Rightarrow \mathbf{B}_{ep1,i} = -\frac{1}{I_{sp} g} \} \]  
(11)

In the case where the propellant mass available to the deputy satellite is fully expelled, the \( \mathbf{B}_{ep} \) input matrix is a zero matrix
\[ \exists \mathbf{x}_{ep} = m_{ip} \{ \mathbf{B}_{ep} = [0 \ 0 \ 0] \} \]  
(12)

Finally, the deputy spacecraft mass \( m_d \), which is a parameter in the NERM model given by Equation 4, can be found using the following formula:
\[ m_d = m_{dry} + m_{ip} - \mathbf{x}_{ep} \]  
(13)

wherein \( m_{dry} \) denotes mass of the deputy satellite without available propellant.

5. Augmented model

The final model is formulated by the augmentation of the principal relative motion model given by Equation 4 using the mass model described by Equation 9. Assuming an augmented state vector:
\[ \mathbf{x}_m = \begin{bmatrix} \mathbf{x}_{rm} \\ \mathbf{x}_{ep} \end{bmatrix} \]  
(14)

the augmented, continuous state-space model of relative motion is given by:
\[ \dot{\mathbf{x}}_m(t) = \begin{bmatrix} \mathbf{A}_{rm} & 0_{6,1} \\ 0_{1,6} & \mathbf{A}_{ep} \end{bmatrix} \mathbf{x}_m(t) + \begin{bmatrix} \mathbf{B}_{rm} \\ \mathbf{B}_{ep} \end{bmatrix} \mathbf{u}(t) + \begin{bmatrix} \mathbf{V}_{rm} \\ \mathbf{V}_{ep} \end{bmatrix} \]  
(15)

where \( \mathbf{V}_{ep} = 0_{1,1} \).

References


Numerical and experimental study of the car aerodynamics

Artur Fityka¹, Arkadiusz Ryfa², Łukasz Walencki¹, Zbigniew Buliński³, Wojciech Adamczyk⁴*

¹,²,³,⁴ Institute of Thermal Technology, Silesian University of Technology
Konarskiego 22, 44-100 Gliwice, Poland
e-mail: arkadiusz.ryfa@polsl.pl

Abstract

The influence of vehicle aerodynamics on its performance and running cost is significant. Car aerodynamics plays crucial role in automotive sport and urban cars. Improvement of urban vehicle bodywork geometry considerably affects the running cost, i.e. the fuel consumption. Prototyping (testing the model in a wind tunnel) is a very important stage during car design, however such measurements are in general very expensive. In order to reduce the cost of prototyping process the Computational Fluid Dynamics (CFD) is used very often. The choice of the best model, which describes the fluid flow (a turbulent model) near bodywork is an important step during numerical modelling of flow around the vehicle. However, these models still require a validation based on reliably measured data. In the paper, the results of numerical simulations were obtained with two different turbulent models solving the flow around bodywork. The results were compared to the measured data, which were obtained from a measuring station. The validation of the model was based on recorded velocity vectors with turbulent energy, which was registered using Particle Image Velocimetry (PIV). The results of numerical simulations are satisfactory, comparable to the experiment data. Moreover, the results indicated areas that should be modified which confirm the usability of computational tools for car aerodynamics improvements.

Keywords: aerodynamics, CFD, PIV, turbulences

1. Introduction

The influence of vehicle aerodynamics on its performance and running cost is significant. Car aerodynamics plays a crucial role in automotive sport. The influence of aerodynamics is also visible on urban cars construction. Improvement of urban vehicle bodywork geometry considerably affects the running cost, i.e. the fuel consumption.

One of the crucial steps in the path of new car development process is the prototyping stage where the car aerodynamic features are investigated in a wind tunnel. Due to high maintenance cost of the wind tunnel operation this manufacturing step is limited only to several tests. In order to reduce the cost of prototyping process the Computational Fluid Dynamics (CFD) can be applied. The choice of the best numerical model and its settings, which describe the fluid flow (a turbulent model) near bodywork are important steps in numerical modeling of flow around the vehicle. Nevertheless, due to many empirical constants used by the numerical model, the validation step based on reliable measured data is still required. In the presented paper, the set of numerical simulations was carried out applying two different turbulent models solving the flow around bodywork. The numerical results were compared against measured data, which were obtained from an experimental rig. The validation of the model was based on recorded velocity vectors and turbulent energy, which were calculated based on recorded flow field using Particle Image Velocimetry (PIV) technique. The results of numerical simulations are satisfactory, comparable to the experimental data. Moreover, the results indicated areas that should be future modified in order to improve vehicle aerodynamics parameters.

2. Experimental rig

In order to validate numerical models and check the aerodynamic feature of the proposed car model the scaled wind tunnel was built at the laboratory of Institute of Thermal Technology. The wind tunnel working conditions give possibility to measure car bodywork aerodynamics in air velocity up to 70 m/s. The wind tunnel is shown in Fig. 1. The flow within the tunnel was stabilized by mounted on tunnel section an appropriate plate with micro orifices section. Moreover, at the end of the wind tunnel additional converging nozzle was installed to stabilized flow pattern. A car model was placed on a platform right behind the outlet.

Figure 1: Experimental rig

The PIV technique was used to measure the flow field around the car. The measurement set was composed of a high-speed camera recording 1000 fps, green laser power of 18 W and an optical kit to regulate the width and length of the laser beam. The velocity of the air outflowing from the tunnel was measured in nine points evenly redistributed over outflow surface. A digital anemometer DO2003 Delta OHM was used to measure the gas velocity. Velocity values measured at nine points were averaged and used as boundary conditions in numerical simulations. The cross section of the wind tunnel area at the outlet was equal to 500x260 mm, so the wind tunnel can

*Acknowledgments. The research was funded within project "Competent mechanical engineers for energetic sector" in POKL.04.01.02-00-131/12.
be used only for small car geometries. In the presented work the set of experiments was carried out in order to validate, selecting turbulence model, as well as for selecting numerical mesh parameters. This data can be used in the future for real car optimization process from aerodynamics point of view. The experiments and numerical simulations were performed for two average velocities equal to 0.85 m/s and 1.94 m/s, respectively to check the experimental usability.

3. Numerical model

Commercial software Ansys Fluent was used to run numerical simulations. Geometry of vehicle with main dimensions are illustrated in Fig. 2.

![Figure 2: Geometrical model](image)

Only half of the vehicle was modeled to reduce number of computational elements, and thus to reduce the time of numerical simulation since geometry and boundary conditions are symmetrical. The number of mesh elements has to be heuristically selected. In this case, the simulations were made for three meshes. Mesh number 1 was made without boundary layer near the bodywork. A total number of grid elements was equal to 8M. For the second mesh the boundary layer was generated. This operation gave possibility to reduce number of elements to 2M. Generated mesh ensured that the distance between body layer and centre of the first numerical element was below one (y+). The last investigated mesh is a middle variant, because it includes a thick boundary layer as well as dense mesh behind the boundary layer. The number of grid elements for last mesh was equal to 7M.

The numerical simulations were made with two models that solve turbulences over the bodywork. As the first one the one-equation Spalart-Allmaras model [1] was selected. This model uses transport equation with modified eddy viscosity for modelling eddies. This quantity is used to determine turbulent dynamic eddy viscosity. This model was designed for aerospace applications. The second model was the Transition SST [2] (shear-stress transport). Transition SST connects k-omega SST model with two additional equations which take into account transition between low and high Reynolds number. K-omega SST was designed for solving both the near-wall region (low Reynolds numbers) and the near boundary layer region (standard k-epsilon model for high Reynolds numbers). Because of its multi-equation the model is very expensive numerically. Furthermore, basic transport equations, continuity equation and three momentum equations (for each dimension) were used. All equations were solved using the second order upwind discretization.

4. Numerical results

The results were compared to the measured data in order to examine numerical mesh and turbulence model influence on numerical simulations result. In Fig. 3 the comparison between the measured data and the results of numerical simulations is done. The data comparison was made at the symmetry plane. Only one case was illustrated here, where the simulation were carried out using SST Transitions turbulent model and mesh with extended boundary layer. The presented results were obtained for 0.85 m/s, and they are comparable to measured data. The aerodynamic drag (Cx) for presented case was equal to 0.25.

![Figure 3: Geometrical model](image)

5. Conclusions

The built experimental rig combined with the PIV technique gives possibility to measure aerodynamics parameters of the scaled car models. The developed numerical model with evaluated mesh parameters and appropriate turbulent model can be used to predict flow field around car. A well-defined numerical model can be used for car prototyping process. Based on the numerical simulation carried out some regions were observed, which can be considered for modification.

References


Static and dynamic accidental load analysis of Jet Hoods

Leszek Flis*
Mechanical Engineering Department, Gdynia Polish Naval Academy
Smidowicza 19, 81-103 Gdynia, Poland
e-mail: l.flis@amw.gdynia.pl

Abstract

The article presents the results of the computer simulations of the Jet Hood units loaded from pressure and caused by an accidental explosion. The Jet Hood is used on oil rigs and must meet the requirements of classification society the Det Norske's which denoting the explosion resistance level. Calculations presented in this paper are an example of the computer simulation methods in practice. The simulation shows a scenario of the calculations that is required and approved by classification societies in case of constructions exposed to explosion. The work paid attention to the scientific threads that need to be taken into account for the correct calculation.

Keywords: jet hood, offshores, blast

1. Introduction

The Jet Hood (type Conex) is designed to extract air from ventilation outlets. Exhaust air is leaded upwards through the unit, and mixed into the atmosphere at the top of the unit. With type Conex the intrusion of water is prevented into the duct systems when ventilation systems are closed down (Fig. 1). The greatest diameter of the device in question is Ø 1750 mm and its height 3250 mm. All details are made from stainless steel AISI 316L sheet with a thickness of 3 mm.

![Figure 1: The Jet Hood unit a) real view  b) surface CAD model in next step converted to FEM shell mesh](image)

2. Computer simulation

The Finite Element Method and explicite LS-DYNA solver was used for the calculations. The mesh model was prepared in HyperMesh preprocessor. The considered structures (Fig. 1) in accordance with the provisions of [1] should withstand a static pressure of 0.2 bar and a dynamic pressure of 0.1 bar. As a general requirement flying objects origin from any explosion load is avoided. For dynamic analysis it is assumed that the over pressure and drag pressure shall be of triangular form, as illustrated in the Fig. 2. The defined $t_1$ is the duration of the positive pulse and $t_2$ is the duration of the negative pulse. Following the over pressure peak loads a negative load due to contraction/reversal of the flow field is specified ($p_1$ is the magnitude of the positive pulse and $p_2$ is the magnitude of the negative pulse). The $t_1$ value is set to 0.2 second pulse duration. In this revision of the only the positive pressure pulse was considered.

![Figure 2: Design explosion pulse](image)

In the modelling by means of Finite Element Method a constitutive equation is one of the most significant elements, since its task is to describe material properties. The Johnson-Cook [2] constitutive equation is employed to resolve numerical problems in which we encounter questions connected with high speed, with high strain rate and with high plastic strain what takes place while blast explosion issue.

$$
\sigma = (A + B \cdot \varepsilon^\nu) \left[ 1 + C \cdot \ln \left(\frac{\varepsilon}{\varepsilon_0}\right) \right] \left[ 1 - (\varepsilon^\nu)^m \right]
$$

where:
- $\sigma$ – von Mises flow stress,
- $A$ – yield stress,
- $B$ – effects of strain hardening,
- $\nu$ – equivalent, effective plastic strain,
- $n$ – exponent strengthening,
- $C$ – strain rate constant,
- $\varepsilon$ – strain rate,
- $\varepsilon_0$ – threshold strain rate,
- $\varepsilon^* = \frac{\varepsilon}{\varepsilon_0}$ – dimension less plastic strain rate,
- $T^*$ – homologous temperature,
- $m$ – temperature exponent.
Assigning Johnson-Cook equation coefficients

Coefficients \( A, B, C, n, m \) from could be assigned, for example, from the results of an experimental test for tensile strength or rotational torque. The first step in assigning the above coefficients is its assigning from the first part of Eqn (1), i.e. \( \sigma = (A + B \varepsilon^n) \), where coefficient \( A \) is the yield stress, and \( B \) and \( n \) represent the effect of strain hardening. If coefficient \( A \) represents the yield stress \( R_y \) in static tensile strength, then \( \varepsilon_0 \approx 1.0 \times 10^{-3} \). In another case, the coefficient \( \varepsilon_0 \), as a baseline, has to mark this strain rate, with which the tensile test diagram was assigned. In the case of the Taylor test, where speed is about 100–400 m/s, the coefficient \( \varepsilon_0 \approx 1.0 \times 10^{-3} \), thus
\[
A = R_y
\] (2)
If the first part of Eqn (1) is rewritten as
\[
ln(\sigma - A) = ln(B) + n \cdot ln(\varepsilon)
\] (3)
and a set of equations with two unknown quantities are solved we are able to assign coefficients \( B \) and \( n \). We could also assign coefficients \( B \) and \( n \) from the formula
\[
n = \frac{\varepsilon_0}{\varepsilon_{TRS}} \frac{\sigma_{TRS}}{(\sigma_{TRS} - A)(1 - \frac{\varepsilon_0}{\varepsilon_{TRS}})}
\] (4)
and
\[
B = \frac{\sigma_{TRS} - A}{\varepsilon_{TRS}^n}
\] (5)
where: \( \sigma_{TRS} \) – ultimate stress, \( \varepsilon_{TRS} \) – ultimate strain at point \( \sigma_{TRS} \), \( E \) – Young’s modulus.

The next step is to assign coefficient \( C \), which represents strain rate. In literature [3], coefficient \( C \) was directly assigned from the diagram as a match of a straight line on a half-logarithmic scale for stand-alone variables, e.g. strain rate. The impact of strain rate and thereby coefficient \( C \) of the next part of the equation
\[
\sigma = (A + B \cdot \varepsilon^n) \left[ 1 + C \cdot \ln \left( \frac{\varepsilon}{\varepsilon_0} \right) \right]
\]
could be assigned by transforming the above equation to form
\[
C = \frac{\sigma - (A + B \varepsilon^n)}{\ln(\varepsilon/\varepsilon_0)}
\]
(6)
The last step is stating the third part of Eqn (1) by means of assigning the temperature exponent \( m \). In order to do so we need information about material strength at room temperature \( T_1 \) and at temperature \( T_2 \) to which material preheats during adiabatic heat exchange. In the same condition at different temperature strengths the proportion is [4]. The coefficient \( m \) could be assigned by transforming Eqn (1) to the form
\[
m = \frac{(\sigma - (A + B \varepsilon^n)) (1 + C \cdot \ln(\varepsilon/\varepsilon_0))}{\ln(1 - r^\alpha)}
\]
(7)

3. Conclusions

The main calculations were performed for a dynamic pressure load 0.2 bar. Figure 4 shows that in this case the Von Mises equivalent stress [MPa] not exceeding stress yield limit.

Additional calculations were performed for dynamic pressure load 0.3 bar. In this case the Von Mises equivalent stress [MPa] exceeding stress yield limit and the unit was collapsed but without defragmentation, what was the main requirement of [1].

References

Abstract

On the basis of potential method the integrated equations of the axisymmetrical edge task heterogeneous thermoelasticity are constructed. The algorithm of the numerical solution of a task is developed. A FORTRAN program is made and the test example is performed. Reliability of an algorithm is checked by means of comparing the received results with the known analytical decision.

Keywords: axisymmetrical body, a potential method, temperature, pressure, thermoelasticity

1. Introduction

As is well-known, while producing the elements of mechanisms, reducing their materials consumption alongside with securing durable and rigid characteristics are of paramount demand. Creation of effective methods of research of SSS (stress-strain state) elements [1-3, 5] that in real conditions are demand. With high temperatures affecting bodies, a change in temperature leads to a significant change of Young module E=E(T), coefficient of linear expansion α=α(T) with the constant Poisson factor ν [3-5].

The purpose of work is development of a potential method for the solution of non-uniform axisymmetric problems of thermoelasticity at variable physimechanical constants.

2. Problem statement. A solution technique

For the sake of examination of SSS in axisymmetrical bodies with continuous heterogeneity it is necessary to set a regional task of the resilience theory (thermo-resilience) and to bring to solving axisymmetrical task of stationary thermoelasticity at variable physicomechanical constants. Using a perturbation method [2, 3], the regional task (1), (2) is brought to solving axisymmetrical task of stationary homogenous body thermo-resilience (zero approximation) and succession of regional tasks of resilience theory (subsequent approximations).

According to [3, 5] the regional axisymmetrical task of heterogeneous thermal resilience is brought to solving differential equphese equations of in particular derivative:

\[ \Delta u - \frac{u}{\rho^2} + 1 \cdot \frac{\partial e}{\partial \rho} - \frac{2(1 + \nu)}{1 - 2\nu} \frac{\partial}{\partial \rho} \left( \alpha(T) \right) = 0 \]

where \( u, w \) – transference, \( \rho, \theta, \vartheta \) – cylindrical ordinates, \( e = \frac{\partial u}{\partial \rho} + \frac{\partial w}{\partial \rho} \), and regional conditions

\[ \frac{\partial}{\partial \rho} + 3v \left( \frac{\partial u}{\partial \rho} \right) = 0 \] \( \frac{1}{2} \left( \frac{\partial u}{\partial \rho} + \frac{\partial w}{\partial \rho} \right) \) \( n \) - directional cosines of outer perpendicular to the surface of the body, and heat conductivity equations too

\[ 1 \frac{\partial}{\partial \rho} \left( \frac{\partial T}{\partial \rho} \right) + \frac{\partial T}{\partial z} = 0 \] \( \frac{\partial T}{\partial z} \) \( n \) - direction cosines of outer perpendicular to the surface of the body, and heat conductivity equations too

where \( T \) – heat conductivity coefficient, \( k \) – empirical coefficient.

Using a perturbation method [2, 3], the regional task (1), (2) is brought to solving axisymmetrical task of stationary homogenous body thermo-resilience (zero approximation) and succession of regional tasks of resilience theory (subsequent approximations). Representing the temperature \( T \) as Green function, instead of (3) we get an integral equation of regional heat conductivity task

\[ 2\pi T(x) = \int_{z} dT(y) \rho d\rho \frac{4}{\sqrt{R^2}} + \int_{z} T(y) \rho d\rho \frac{2}{\sqrt{R^2}} \times \left\{ \frac{2\rho}{r_0} \left( \rho_0 - \rho \right) \right\} \] (4)

Here \( r, y \) – parametrical and flowing points at integration, \( r^2 = \rho^2 + \rho_0^2 - 2\rho_0\rho \cos Q + Z^2; \ Z = Z_2 - Z_1; \ Q = \theta - \theta_0; \ dl_r \) – the element of the meridional contour, \( Z, E, K \) – elliptical integrals. The solution of system (1) is sought in the form of

\[ u_\rho = u_{\rho_0}^0 + u_\rho^1, \quad u_z = u_z^0 + u_z^1, \] (5)
where \( u^0 \), \( u^i \) - general solutions of similar differential equations (1), which are given in [1], and \( u^0, u^i \) - Goodier’s particular solutions, the integral representation of which looks like [1].

With the help of Duhamel-Neuman correlations were built integral equations of strains \( \sigma_{rr}, \sigma_{zr}, \sigma_{zz}, \sigma_{rz}, \) (corresponds to (5), and expressions \( \sigma_{rr}, \sigma_{zr}, \sigma_{zz}, \) are given in [1, 5]) and singular integral equations (SIE) of regional task of thermoresilience at zero approximation

\[
v_r(x) = \frac{1}{4\pi(1-v)} \int v_r(y) \left[ (A_{rr} n_r + A_{rr} n_r) + (A_{zr} n_r + A_{zr} n_r) \right] dy + v_r(y) B_{rr} n_r dy + p_r(x) + p_r^f(x),
\]

\[
v_z(x) = \frac{1}{4\pi(1-v)} \int v_z(y) \left[ (A_{rr} n_r + A_{rr} n_r) + (A_{zr} n_r + A_{zr} n_r) \right] dy + v_z(y) B_{rr} n_r dy + p_z(x) + p_z^f(x),
\]

where \( v_r, v_z \) - simple layer potential densities, \( A_{rr}, ..., B_{rr} \) - coefficients received in [1], \( p_r(x), p_z(x) \) - components of mechanical strains, \( p_r^f(x), p_z^f(x) \) - components of fictitious temperature surface load.

Here the solution of the resilience theory task is found as

\[
p_r^f(x) = - \left( \sigma_{rr} - n_r + \sigma_{zz} - n_z \right),
\]

\[
p_z^f(x) = - \left( \sigma_{rr} - n_r + \sigma_{zz} - n_z \right).
\]

As a result of the solution (6) were defined the densities \( v_r, v_z \), and then the strains \( \sigma_{rr}, ..., \sigma_{zz} \). The strain values were used for solving the regional resilience theory task at first approximation.

Here \( u^0 = u^0 + u^i, u^i = u^i + u^i \).

Integral presentations \( u^0 \) and \( u^i \) are known [1, 5], and

\[
u_r^0 = \frac{1}{4\pi(1-v)} \int f(T) \left[ \frac{X_r E(T)}{C_{rr}} + \frac{X_z E(T)}{C_{zz}} \right] dS_r,
\]

\[
u_z^0 = \frac{1}{4\pi(1-v)} \int f(T) \left[ \frac{X_r E(T)}{C_{rr}} + \frac{X_z E(T)}{C_{zz}} \right] dS_z,
\]

where

\[
f(T) = \frac{1}{E} \frac{dT}{dt}, \quad X_r = \left( \frac{\partial^2 T}{\partial r^2} \sigma_{rr} + \frac{\partial^2 T}{\partial r \partial z} \sigma_{rz} \right),
\]

\[
X_z = \left( \frac{\partial^2 T}{\partial z^2} \sigma_{rr} + \frac{\partial^2 T}{\partial r \partial z} \sigma_{rz} \right) C_{rr} \sigma_{rr} + C_{zz} \sigma_{zz} - \text{similar to the correlation in the paper [1, 5].}
\]

The SIE system is the same as in (6), but instead of \( p^f \) is \( p^f \), a fictitious load is taken

\[
p_r^f = - \left( \sigma_{rr} - n_r + \sigma_{zz} - n_z \right), \quad p_z^f = - \left( \sigma_{rr} - n_r + \sigma_{zz} - n_z \right).
\]
The impact of fire situation on the static and stability response of the bearing steel structure

Andrzej Garstecki¹, Katarzyna Rzeszut², Łukasz Polus¹, Maciej Klój⁴, Mateusz Terech⁵

¹Polytechnic Institute, Stanisław Staszic University of Applied Sciences in Pila
Podchorazych 10, 64-920 Pila, Poland
e-mail: andrzej.garstecki@put.poznan.pl

²,³,⁴,⁵ Faculty of Civil and Environmental Engineering, Poznań University of Technology
Piotrowo 5, 60-965 Poznan, Poland
e-mail: katarzyna.rzeszut@put.poznan.pl, lukasz.polus@put.poznan.pl, klojmaciej@gmail.com, mateusz.terech@student.put.poznan.pl

Abstract

In the article, the behaviour of multiple bay portal frame structures in fire conditions are analysed. The analysed single-storey building has two bays and one fire wall which separates the bays. Only one of the bays has fire protection. The analysis is carried out for fire accidental action using the standard ISO-fire curve. The iterative procedure to investigate critical temperature is used. At each increment of temperature the ultimate limit state is checked using the EC recommendation. Special attention is focused on failure analysis. First, using the public domain program LUCA the structure is checked if it does not collapse towards the outside of the building in case of fire occurring in one of the building compartments. The criteria which guarantee the safety of occupants and firemen are considered. Secondly, using the same program it is checked if the localized failure of the unprotected compartment leads to the collapse of the protected compartment. At this stage of analysis the additional tensile forces and lateral displacement resulting from the collapse of unprotected compartment are calculated. The conducted analysis leads to the improvement of structural safety. It must be underlined that the impact of the fire situation can be significant, enforcing important changes in the project. Therefore in the design of steel structures fire protection and the behaviour of the structure during fire must be taken into account.

Keywords: single storey-buildings, structural fire analysis, fire situation, critical temperature

1. Introduction

Nowadays, fire design develops very fast and there are more and more methods which improve design process. Storage and industrial buildings constructed with a steel structure in a fire situation may be analysed using several simplified design methods. These methods allow the designer to evaluate easily the mechanical behaviour of steel structure exposed to high temperature [1]. However, there is still deficit of examples which present application of these simplified methods with proper conscience of real situations and eventual need of using more sophisticated analyses. One of the earliest European regulations including fundamental design rules for single-storey steel buildings accounting for fire conditions are provided in [4] and [2]. Some interesting fire resistance analyses of simple steel frame were presented in [3].

2. Problem formulation

The most common way to protect single-storey steel buildings against fire is the use of firewalls and fire protection of the steel elements. Firewalls divide buildings into fire compartments which limit damage only to this compartment where the fire occurred (see Figure 1 and 2). However, the localized failure of the unprotected compartment in fire situation may leads to the collapse of the neighbouring bays [4]. The most dangerous situation for firemen is when the structure collapses towards the outside. For this reason, it is necessary to check if it can occur. However, the main purpose of this paper is to analyse localized failure of the unprotected compartment in fire condition and its influence on the other parts of the building. This influence may be evaluated using simplified methods implemented in public domain programme Luca. It makes possible the determination of forces which are generated...

Figure 1: Partition of the single-storey building into fire compartments

Figure 2: Forces caused during collapsing of the structures of the compartment in fire
by the collapsing structure in the compartment area covered by the fire. These forces are used as additional horizontal load during the structural analysis in normal load condition at the room temperature. Moreover, using simplified methods, it is possible to evaluate maximum horizontal displacements and check whether fire walls or building facades will be destroyed or not.

3. Tensile forces and lateral displacement at the fire compartment

The horizontal tensile force at the fire compartment boundaries may be determined using the recommendations implemented in [4] and the formula:

\[ F = c_p n_{eff} q l \]  

where:
- \( c_p \) is an empirical coefficient depending on the slope of the roof and the type of steel structure
- \( n_{eff} \) is a coefficient related to the total number of heated bays in the fire compartment
- \( q \) is the linear load on roof applied on the beam in the fire situation [N/m]
- \( l \) is the span of heated bay connected to the column [m]

Maximum lateral displacements at the top of columns \( \delta_i \) (\( i = 1, 2 \)) located at the compartment boundary (Figure 3) can be obtained from:

\[ \delta_i = \frac{K_i c_i n l}{K} \text{ fire at the end of the building} \]
\[ \text{Max} \left\{ \frac{K_i c_i n l}{K}, \frac{F}{K} \right\} \text{ fire in the middle of the building} \]  

where:
- \( n \) is the number of heated bays
- \( K_i \) is the equivalent lateral stiffness of the considered part \( i \) of the structure [N/m]
- \( K \) is the equivalent stiffness depending on equivalent stiffness \( K_1 \) and \( K_2 \) [N/m]
- \( l \) is the span of the heated bay connected to the column
- \( F \) is the tensile force [N]
- \( c_i \) is an empirical coefficient dependent on the slope of the roof and type of steel structure.

4. Calculation examples

In the paper two numerical examples are presented. In each case the structure consists of two bays steel frame but in one the truss and in second the welded girder is applied. The geometry of two steel frames are presented in Figure 4. In both examples, the fire wall is located in the middle of the hall and fire burst up only in one of the compartments.

Next, these forces are used as additional horizontal load in structural analysis in normal load condition at the room temperature. In both examples, additional horizontal forces led to significant lateral displacement and firewall destruction. For this reason, the overall constructional stiffness should be improved and the stiffness of the columns should be increased.

5. Conclusions

The conducted analysis showed that fire of the unprotected compartment of the multi-bay hall can cause the collapse of the steel roof structure within the bay engulfed by fire. It induced additional tensile force which affected the neighbouring bays. As a result, large displacements occurred at the top of the columns and damaged the firewalls. In order to avoid it the stiffeners of the column should be increased but in some cases it can be uneconomical. Therefore an alternative way is to protect both bays at the same fire resistance level or divide the structure into two separate bays with double columns in the middle.

References

Form finding of tensegrity structures via singular value decomposition of compatibility matrix

Wojciech Gilewski¹, Joanna Kłosowska², Paulina Obara³

¹ Faculty of Civil Engineering, Warsaw University of Technology
Al. Armii Ludowej 16, 00-637 Warsaw, Poland
e-mail: w.gilewski@ilt.pw.edu.pl

²,³ Faculty of Civil Engineering and Architecture, Kielce University of Technology,
Aleja Tysiąclecia Państwa Polskiego 7, 25-314 Kielce, Poland
e-mail: Joanna.Klosowska@interia.pl, paulina@tm.kielce.pl

Abstract

Application of singular value decomposition (SVD) of a compatibility matrix to form-finding of pin-joined structures is presented in this paper. This decomposition allows identification of the mechanism of geometrical variation of a truss, its preliminary classification as finite or infinitesimal and determination of longitudinal forces which respond to the self-stress state. Calculations were made in the Mathematica environment. The computational program for tensegrity form-finding and qualitative verification of truss properties was written based on the finite element analysis within the Mathematica environment.

Keywords: singular value decomposition, truss, tensegrity, self-stress state, mechanism

1. Introduction

The idea of tensegrity was first described about 50 years ago. The concept concerns specific trusses, which consist of compression and tensile components which stabilize each other despite the fact that there are mechanisms in the structures. A qualitative analysis of pin-jointed structures was carried out in this paper. It was made by means of the decomposition matrix method which describes the elongations in a truss according to singular values (SVD) [2,3]. This method allows to state if the structure is infinitesimally geometrically variable and whether there are self-stress states.

In the SVD decomposition a given matrix is presented in the form of a product of the unitary square matrix, the rectangular diagonal matrix with non-negative real coefficients and the Hermitian conjugation of a unitary square matrix. The coefficients of the diagonal matrix are called singular values of the analyzed matrix. When the given matrix includes real coefficients the unitary matrices become orthogonal matrices and the Hermitian conjugation becomes a transposition. The paper presents the essence of a singular value decomposition of the matrix and an example, explaining what information about the structure can be obtained according to this distribution. SVD of compatibility matrix of truss theory has been hinted at in literature, usually for statically determinate structures.

2. Governing equations

The subject of the truss analysis consists of N members which are, unloaded, supported, with following characteristics: material constants $E_e$, cross-sectional areas $A_e$ and bar lengths $L_e$ ($e = 1, \ldots, N$). Its mechanical properties are described by three linearized equations: compatibility, material properties and equilibrium with boundary conditions included:

$$\Delta = B \varepsilon,$$
$$S = E A,$$
$$B^T S = P,$$

where $\varepsilon$ is displacement vector, $B$ is compatibility matrix, $A$ is extension vector, $S$ is internal bar tensions, $E$ is elasticity matrix and $P$ is load vector.

The compatibility matrix $B$ of an analyzed truss can be determined directly or using the formalism of the finite element method [1].

3. Singular value decomposition

The singular value decomposition of an $N \times M$ real matrix $B$ is a factorization of the form:

$$B = Y N X^T$$

where $Y$ is an $N \times N$ real orthogonal matrix, $X$ is an $M \times M$ real orthogonal matrix and $N$ is an $N \times M$ rectangular diagonal matrix. Let us consider two eigenproblems and their solutions.

$$\begin{align*}
B^T B - \mu I &= 0, \\
B^T B - \lambda I &= 0,
\end{align*}$$

with the solutions in the form of eigenvalues and eigenvectors

$$\begin{align*}
\mu_1, \mu_2, \mu_3, \ldots, \mu_N, \\
\lambda_1, \lambda_2, \lambda_3, \ldots, \lambda_M,
\end{align*}$$

Full solutions of the above eigenproblems can be expressed in the condensed form

$$BB^T = YY^T,$$

$$B^T B = XLX^T,$$

where

$$M = \text{diag} [\mu_1, \mu_2, \ldots, \mu_N],$$
$$L = \text{diag} [\lambda_1, \lambda_2, \ldots, \lambda_M],$$
$$Y = [y_1, y_2, \ldots, y_N],$$
$$Y^T Y = I.$$
Note that the product $BB^T$ can be considered as a matrix of symmetrised equations of equilibrium with non-negative eigenvalues. The zero eigenvalues (if any) are related to the non-zero solution of homogeneous equations ($P=0$) named self-stress. The self-stress can be considered an eigenvector related to zero eigenvalue).

In a similar way the product $B^TB$ can be considered a particular form of linear stiffness matrix with unit elasticity matrix. The eigenvalues are non-negative. Zero eigenvalues (if any) are related to the finite or infinitesimal mechanisms, but in general the information from the null-space analysis alone does not suffice to establish the difference. The mechanism can be considered an eigenvector related to zero eigenvalue). In order to find out if the mechanism is infinitesimal it is necessary to apply nonlinear analysis with the use of geometric stiffness matrix, which is possible if the self-stress exist. The lack of self-stress means that the mechanism is finite. Based on the above two eigenproblems it is easy to prove the Singular Value Decomposition of the Compatibility matrix $B$:

$$BB^T = YN^TV^T Y = YMY^T,$$

$$B^TB = XN^TV^T XN = XIX^T,$$

with the following relations included:

$$M = NN^T,$$

$$L = NN^T.$$  

4. Example

The paper explains what kind of information about a structure can be obtained from the singular value decomposition of the compatibility matrix $B$ (4). Calculations were made in the Mathematica environment. In order to demonstrate the method, the tensegrity truss named Simplex (Fig. 1) is presented. The vectors of generalized coordinates and longitudinal forces are: $q = \{q_1, q_2, ..., q_{12}\}; S = \{S_1, S_2, ..., S_6\}$.

For this case $B$ is the 12x12 real matrix. The square roots of both $BB^T$ and $B^TB$ matrices are found in the diagonal entries of $N$. One of the eigenvalues is zero.

The mechanism is presented in Fig. 2. There is a single self-stress observed in the structure with the following normalized forces (Fig. 1): top and bottom cables ($1,2,3,4,5,6$) 0.17, vertical cables ($10,11,12$) 0.3 and vertical struts ($7,8,9$) -0.43. The mechanism can be compressed by the self-stress state to build tensegrity.

5. Conclusions

The paper discusses the use of a singular value decomposition for the qualitative analysis of trusses, including tensegrity structures. The analysis allows one to draw the following conclusions:

- the knowledge of the compatibility matrix $B$ is sufficient to analyze the qualitative properties of a truss, including the configuration of nodes and rods. Therefore only the geometric description is required. It is not necessary to know the cross bars and the material properties,

- the singular value decomposition allows identification of the mechanism of geometrical variation of a truss and its preliminary classification as finite or infinitesimal,

- the singular value decomposition allows determination of the normal forces which respond to the self-stress state, if there are any in the analyzed truss.

The consideration set out in the work can be used to analyze both two- and three-dimensional trusses.

References


Truss model of origami inspired folded structures

Wojciech Gilewski¹, Jan Pelczyński²

¹,²Faculty of Civil Engineering, Warsaw University of Technology
Al. Armii Ludowej 16, 00-637 Warsaw, Poland
e-mail: w.gilewski@il.pw.edu.pl, j.pelczynski@il.pw.edu.pl

Abstract

Origami is an old art of paper folding. From mechanical point of view origami can be defined as a folded structure. The paper shows an attempt to build a simplified model of origami-inspired folding structures based on the pin-joined space bars, which allows to analyze the influence of the introduced geometry on the global properties of the structure. Eggbox and Miura Ori origami modules are used in the analysis.

Keywords: origami, folded structure, truss model

1. Introduction

Folded plates are attractive solutions for engineers. There are a lot of interesting engineering structures of this kind, some of them are based on the concept of origami - an old art of paper folding dated for the 7th century [1]. From mechanical point of view the correct theory to describe folded plates is a six parameter shell theory with three displacements and three rotations in the displacement field [2,4]. The third rotation is necessary due to the folding of the structure. Each fold consists of flat surfaces, so the equations of the theory can be simplified. For numerical analysis it is necessary to use the finite element method because of the complex character of the structures. Finite elements with six d.o.f. per node are to be used. However, shell models of folded structures provide the models with a lot of degrees of freedom and need application of professional software. The most challenging task is to develop an effective technique for efficient computation of structures with a lot of folds [4].

The present paper is an attempt to create a simplified model of origami-inspired folding structures based on the pin-joined space bars. The aim is not the miniautie of the stress distribution. The aim of the model is to obtain different information – to analyze the effect of the introduced geometry on the global properties of the sheet. Dominant mechanisms and self-stress states – the internal normal forces states, which satisfy the homogeneous equilibrium equations – are considered.

2. Origami inspired structures in engineering

Origami is an old art of paper folding developed in Japan with origins in China. The term is a combination of two Japanese words: 'oru’ - to fold and ‘kami’ - a paper. There are several typical folds in the origami art: a ‘mountain fold’, a ‘valley fold’ and a ‘swivel fold’. After changing the direction of the mountain fold one can receive an ‘inside reverse fold’ or an ‘outside reverse fold’. The other possibilities are a ‘squash fold’ and a ‘sink fold’. A detailed description of origami folds and patterns can be found in [3]. An example of the origami shape is presented in Fig. 1.

Origami is an inspiration for engineers in the fields of civil engineering, architecture, biotechnology, medicine, space engineering and other technical applications [5–7]. The most attractive areas of civil engineering applications are:

- deployable structures,
- stiff structures with minimal expense of weight,
- shock absorbing devices.

The folded structures can be capable of changing their shape to accommodate to the new requirements, whilst maintaining a continuous external surface.

![Figure 1: Miura Ori origami shape](image)

3. Simplified mechanical model

From mechanical point of view origami can be defined as a folded structure [1,3]. In the present paper the use of a pin-jointed bar framework to represent the origami folding is presented. Its mechanical properties are described by three linearized equations: compatibility (Eqn (1)), material properties (Eqn (2)) and equilibrium (Eqn (3)):

\[ \Delta = Bq \] (1)

\[ S = E\Delta \] (2)

\[ B^T S = P \] (3)

where \( q \) is a displacement vector, \( B \) is a compatibility matrix, \( \Delta \) is an extension vector, \( S \) is an internal bar tensions, \( E \) is an elasticity matrix and \( P \) is a load vector.

Following the singular value decomposition of the compatibility matrix \( B \) it is possible to define the finite or infinitesimal modes of the structure as well as the self-stress states. It gives the qualitative information about the global stability of the structure.

4. Analysis

For further analysis two truss structures, inspired by Miura Ori and Eggbox origami schemes (Fig. 2), were prepared. Both
are presented in Fig. 3. The supports are realized by bars that are simply supported at one end.

![Figure 2: Modules of Eggbox (a) and Miura Ori (b) structures](image_url)

Figure 2: Modules of Eggbox (a) and Miura Ori (b) structures

For each truss the matrices $B$ and $E$ were built. Thereby, by finding the eigensolution of the matrix $B^TB$, it was determined that four infinitesimal modes occur in the analyzed structures. Examples of the rigid movements are shown in Fig. 4. Also the eigensolution of the matrix $BB^T$, which indicates the number of self-stresses occurring in the truss, and the corresponding eigenvectors, which specify the distribution of the self-stress force over bars, were found. Both trusses are characterized by one self-stress state. Figure 5 shows bars under tension and compression in the self-stress state.

The linear stiffness matrix $K_L = B^TEB$ was calculated. In order to eliminate four rigid movements, the stiffness matrix was modified by the addition of the geometrical stiffness matrix $K_G$, which was developed following a standard FEM procedure [8] based on the self-stress of the structure. Further analysis of the stiffness matrix $K = K_L + K_G$ shows that all zero eigenvalues changed to positive or negative values. Although the matrix $K$ is not positive definite it is possible to solve the $Kq = P$ system and obtain reasonable results. The qualitative analysis of the result is the subject of further analysis.

For the truss composed of more than one single modules some results were obtained. Analysis shows that this kind of structure has more infinitesimal modes, but number of self-stress states is constant and equals one. Some results are presented in Fig. 6.

5. Conclusions

Approaches which simplify the process of obtaining information about the global properties of origami-inspired folded structures were proposed. Spectral analysis of the matrices $B$ and $K$ was performed. Thereby infinitesimal modes of the structure were found, which afterwards were eliminated by addition of self-stress states.

The qualitative analysis of structures with self-stress included is the subject of further analysis.

References


Investigation of resonance diffraction by hidden obstacles using laminate element method

Evgeny Glushkov, Natalia Glushkova, Artem Eremin, Rolf Lammering

1,2,3Institute for Mathematics, Mechanics and Informatics, Kuban State University
Stavropolskaya 149, 350040 Krasnodar, Russia
e-mail: evg@math.kubsu.ru
4Institute of Mechanics, Helmut-Schmidt-University/University of the Federal Armed Forces
Holstenhofweg 85, 22043 Hamburg, Germany
e-mail: rolf.lammering@hsu-hh.de

Abstract

A semi-analytical laminate element technique developed for the investigation of peculiarities of guided wave propagation and diffraction in layered structures with local inhomogeneities is presented. The approach is based on the use of fundamental solutions for the pristine layered structure as as basis functions for the scattered field approximation. These solutions identically satisfy the governing equations in the sublayers and all interface and boundary conditions on the plane-parallel surfaces. As an example, its application to the investigation of resonance guided wave interaction with a hidden cavity in an elastic layer is considered. The results of numerical simulation are experimentally confirmed.

Keywords: elastic waveguide, hidden defects, guided waves, laminate elements

1. Introduction

Structural health monitoring of plate-like units is based on the propagation of elastic guided waves (GW) for long distances and their interaction with local inhomogeneities (defects) of various types. One of the most advantageous for the defect detection is a resonance GW diffraction featured by the capturing of incident wave energy and its prolonged localization in the defect vicinity in the form of weakly decaying standing waves at the resonance frequencies. Computer simulation aims to clarify such a complex ultrasonic wave motion. General-purpose and specific finite element computational codes proved their efficiency for calculating wave propagation in elastic structures with a complex geometry [1]. At the same time, due to lengthy laminate element formalisms, algorithms of boundary integral equation (BIE) technique, are often more efficient. It allows to reduce the problem dimension and obtain the results in a physically clear form of GW asymptotic expressions. This technique becomes especially convenient when fundamental solutions for the elastic layered structure are used as kernel functions. Such basis functions, called laminate or layered elements (LEs), identically satisfy the governing equations in the sub-layers, e.g. the interface boundary conditions, and the conditions on the exterior plane-parallel surfaces [2]. Therefore, with such kernels, the integration over the obstacle surface only is enough for a scattered wave representation. All wave properties of the structure with possible high gradients at contrast interfaces are automatically taken into account.

In the paper we demonstrate the application of the BIE approach with LE kernels (further referred to as LEM-BIE) to the investigation of resonance GW interaction with a horizontal tunnel-like cavity in an elastic plate. Experimental validation of the results obtained is performed on an aluminium specimen with an elliptical void. The incident waves are excited by a surface mounted piezoelectric wafer active sensor while a contactless laser Doppler vibrometry (LDV) is utilized for the wave propagation sensing and visualization.

2. Mathematical model

Let us consider a plane-strain time-harmonic oscillation \( u_0 e^{-i\omega t} \), \( u = \{ u_x, u_z \} \) of a layered isotropic linear-elastic plate of thickness \( H \) governed by the Lamé equations. In the Cartesian coordinate system \( x = (x, z) \), the structure occupies the domain \( D = \{ |x| < \infty, -H < z < 0 \} \) with an elliptical void with the surface \( S : (x-x_0)^2/a^2 + (z-z_0)^2/b^2 = 1 \); actually, it is a 2D cross-section of a horizontal hole (Fig. 1). The exterior plane-parallel surfaces \( z = 0 \) and \( z = -H \) and the void’s surface \( S \) are stress free.

A specified incident GW \( u_0 \) travels from left to right. Its diffraction by the obstacle gives rise to the scattered field \( u_{sc} \), so that the total wave field \( u \) is a sum of the known and unknown fields: \( u = u_0 + u_{sc} \). The unknown field \( u_{sc} \) is investigated within the indirect LEM-BIE formalism in terms of the laminate element fundamental 2 \( \times \) 2 matrix \( l(x) \) integrated over the defect boundary \( S \) together with an unknown source density \( c(x) \):

\[
 u_{sc}(x) = \int_S l(x - \xi) c(\xi) d\xi
\]  

(1)

The algorithms of \( l(x) \) semi-analytical calculations for multilayered elastic structures are given in Ref. [2] and works cited therein while the vector factor \( c(\xi) \) is discretized by a boundary element

This work is partly supported by the Russian Foundation for Basic Research (RFBR) under the grants No. 13-01-96516 and 14-08-00370
representation:

$$\mathbf{u}_x \approx \sum_{j=1}^{N} \mathbf{u}_{x,j}(\mathbf{x}) = \int_{S_j} l(\mathbf{x} - \xi) \mathbf{c}_j d\xi, \quad (2)$$

where $S_j$ are segments of a polygon approximation of the smooth boundary $S$; $\mathbf{c}_j$ are vectors of unknown constants. They are obtained from the system of linear algebraic equations with respect to the integrated vector of unknown constants $\mathbf{e} = (\mathbf{c}_1, \mathbf{c}_2, ..., \mathbf{c}_N)$, which arises after the substitution of relation (2) into the boundary conditions on $S$ and further implementation of the Galerkin projection scheme. An efficient numerical evaluation of the multifold path integrals arising here is achieved using the trick based on the specific rotation in the Fourier parameter space bringing multifold path integrals to onefold ones. It allows to use the residue technique, which drastically reduces the computational expenses and gives the far-field GW asymptotic expression for the scattered field $\mathbf{u}_x$.

3. Resonance effects, FEM and experimental validation

The LEM-based parametric analysis revealed the trapping mode effect, mathematically associated with the spectral points $\hat{\omega}_n = 2\pi f_n$ of the boundary value problem considered that are located close to the real axis in the complex frequency plane (see Ref. [3] and papers cited therein). With a sole obstacle, this effect manifests itself in a sharp incident wave screening at the frequencies $f_n \approx \Re f_n$. As an example, Fig. 2(a) demonstrates such sharp minima of the $S_0$ mode transmission coefficient $\kappa^+ (f)$ at the frequencies $f_1 = 102$ kHz and $f_2 = 231$ kHz; the nearest complex spectral points are $f_1 = 103(1 - i0.031)$ and $f_2 = 235(1 - i0.012)$.

$$|v(x, f)|$$

Figure 2: Frequency dependency of the $S_0$ mode transmission coefficient (a), normalized spectra of the out-of-plane velocities LDV measured in the points $C_1$ and $C_2$ (b)

The calculations are carried out for the input parameters corresponding to the aluminium plate sample used in the experiments: $E = 70$ GPa, $\nu = 0.33$, $\rho = 2700$ kg/m$^3$, and $H = 5$ mm; the obstacle is the middle elliptical hole with the semi-axes $a = 3.5$ mm and $b = 1$ mm. The transmission coefficient is introduced as the ratio of the time-averaged wave energy $E^+_{\omega}$ carried by the $S_0$ mode behind the obstacle to the incident wave energy $E_{\omega}$:

$$\kappa^+ = E^+_{\omega} / E_{\omega}.$$  

In accordance with the trapping mode theory, the resonance screening is followed by the localization of oscillations near the obstacle. Indeed, the displacements at the frequencies $f_n \approx \Re f_n$ exhibit such a localization above and beneath the obstacle (Fig. 3 top). The level-line images of Fig. 3 show a spatial distribution of the amplitude $|u_x(x, z)|$ of the vertical displacement component of the corresponding eigenforms. The two lower images are for the validating FEM results obtained using the COMSOL Femlab 3.5a software for the limited rectangular specimen with the void (a finite cut from the infinite waveguide considered). The FEM-based eigenfrequency analysis has revealed a number of eigenforms of that finite sample. But only two of them shown in the figure are featured by the localization of oscillation at the void. These eigenforms $u^E_{x}(x, z)$ and eigenfrequencies $f_n^{FEM}$ are very close to the LEM-based obtained ones.

$$\omega_{n}$$

Figure 3: Localization patterns for the infinite waveguide (a,b) and corresponding eigenforms of the finite specimen (c,d) (dark regions correspond to higher amplitudes)

It is clear that the first eigenform exhibits maximal resonance out-of-plane displacement of the exterior surface just above the void center while the maxima of the second form are shifted aside. Therefore, in order to validate the predicted resonance effect and the calculated resonance frequencies $f_1$ and $f_2$, LDV measurements of the out-of-plane displacement velocity $v_x = u_x$ were accomplished at the points $C_1$ and $C_2$ (Fig. 1).

Similarly to the results obtained for the delaminated plate [3], the vibrometry revealed long-standing-wave-type oscillations above the void. Spectral analysis of that prolonged motion has shown two strong peaks of the velocity frequency spectrum $|v_x(x, f)|$ at the frequencies $f_1^{exp} = 106$ kHz and $f_2^{exp} = 222$ kHz (Fig. 2 bottom). As expected, the first, associated with the first eigenform, manifests in the data recorded at the point $C_1$ and gives no effect in the $C_2$ results. While in agree with the eigenform shapes, only the second resonance frequency $f_2^{exp}$ gives a peak growth of the spectral amplitude for the data acquired from the point $C_2$.

References


Parametric analysis of Istanbul's Ring Road viaduct for three levels of seismic load

Karol Grębowski¹, Michal Hirsz², Adam Nadolny³, Krzysztof Wilde⁴

¹,²,⁴ Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland

e-mail: karol.grebowski@pg.gda.pl¹, mhirsz@pg.gda.pl², krzysztof.wilde@pg.gda.pl⁴

³ Mosty Gdańsk Sp. z o.o.
Jaśminowy Stok 12A, 80-177 Gdańsk, Poland

e-mail: adam.nadolny@mostygdansk.pl

Abstract

The paper presents a parametric analysis of the Istanbul's ring road viaduct that is currently under construction within the Northern Marmara Highway project. The structure, due to its location on seismic prone areas is exposed to seismic loads of different strengths and different return periods. The study is focused on concrete bridge supports that are designed to work in nonlinear range. The parametric study, conducted in MATLAB and SOFISTIK environment, aims at detailed verification of the viaduct's supports performance due to three levels of seismic loads intensity. The research and analysis procedures will be used in recommendations for development of slender bridge supports of variable geometry with stiffness chosen to achieve the best performance to weak and very strong earthquake.

Keywords: concrete viaducts, seismic loads, dynamic analysis, plastic hinges, FEM modelling

1. Introduction

Istanbul is the largest and most populated city in Turkey. Its boundaries stretch on both sides of the Bosphorus, from the northern coast of the Sea of Marmara, to the southern coast of the Black Sea. As one of only two cities in the world, it is located in both Europe and Asia. Istanbul is located near the North Anatolian Fault which includes a place where two tectonic plates, African and Eurasian, meet. As a result, the entire region is one of the most seismically active areas on the Earth. In 1999, in Kocaeli province, a place several dozen kilometers away from Istanbul, the 7.6 on Richter scale earthquake took place. It caused the deaths of almost 18,000 people and led to the destruction of many buildings and bridges that played a very important role in the region's development.

In 2010, due to the development of towns and suburbs of the Black Sea, an idea of building a third ring road of Istanbul within the framework of the Northern Marmara Highway project arose. In 2013 the project started to be implemented. The route includes 67 viaducts, and the length of the designed section is 260 kilometers. This solution became an alternative for the two bridges over the existing ring roads of the city, and is a crucial factor in the development of the industrial sector of the region.

Due to the fact that the Northern Marmara Highway ring road is constructed on the areas with a high seismic activity, a problem of designing the structure of viaducts and bridges, resistant to seismic load occurs. Especially, if one takes into consideration the fact that a study conducted in 2000 showed, there is a 60 percent chance of 7 on Richter scale earthquake will occur in Istanbul by 2013 [1,4].

2. Seismic loads

The Turkish norm DLH 2008 [2] includes three earthquake intensity levels for defining the seismic loads:

- **D1** - return period: 72 years (50% in 100 years)
- **D2** - return period: 475 years (10% in 100 years)
- **D3** - return period: 2475 years (2% in 100 years).

It is assumed the viaduct's structure (Fig. 1) subjected to seismic loads, corresponding to an occurrence of earthquake of level D1 will avoid damaging.

![Figure 1: Concept of variable stiffness of the intermediate supports](image-url)

It is assumed that for earthquakes of level D2 the viaduct's structural elements will develop some cracks, but the damage will occur in controlled areas which might be repaired within a few months period with little traffic restrictions (level D2). The intermediate supports have a variable height due to the geometry of the crossing (Fig. 1). There are large differences in rigidity of the supports depending on the height and cross-sectional geometry. The controlled damage in the bridge structure is intended to be located in the upper part of the supports in the form of developing plastic hinges. The deck should not fall off from the supports at level D3 earthquake [1].

The simulations are based on a seismic load that was recorded during the quake of 17 August 1999 near Istanbul's province Kocaeli (Fig. 2).
3. Parametric Analysis of the V6 viaduct in Matlab and Sofistik environment

In order to perform a parametric analysis of the selected Istanbul's ring road a viaduct V6 was selected as a representative example. The purpose of this analysis is to verify a simplified numerical bridge model derived in Matlab with solution obtained with use of Sofistik. Finally, an extensive parametric study for different concrete supports stiffness and different levels of earthquake excitation have been conducted.

In MATLAB program, a numerical model of the viaduct is a multi-span beam with the intermediate supports modelled by horizontal springs of an appropriate stiffness depending on the type of cross-section and pillars' height (Fig. 3).

Figure 3: Numerical model of V6 viaduct in Matlab

A numerical model derived in SOFISTIK environment of the viaduct consisting of the superstructure and the pillars is shown in Fig. 4.

Figure 4: Numerical model of V6 viaduct in Sofistik

The parametric analysis assumes that the load of Kocaeli scaled to the three required levels of earthquake is applied on the bridge. The best configuration of the pillars geometry (Figure 1) i.e., relation between the stiffness of the bottom of the pillar, \( k_1 \), and the stiffness of the upper part, \( k_2 \), is determined. The selection of the pillars geometry must satisfy the constriction condition and comply with Turkish norm Türkisch DLH 2008 [2,3].

4. Results and final remarks

The example of the results from the linear and nonlinear analysis of displacements in the span center of the viaduct for seismic load level D1 are shown in Figs. 5 and 6. In this example it is assumed that the pillars' upper part is of the same length for all the intermediate supports and is equal to \( l_2 = 18 \text{ m} \).

Figure 5: Linear analysis - Horizontal displacement in the span center of the viaduct for the D1 load level

Figure 6: Simulation from nonlinear analysis of the viaduct for the D1 load level – Sofistik

The linear analysis of seismic loads provided the same results for both models derived in Matlab and Sofistik environment. The maximum horizontal displacement of the bridge deck amounted to 150 cm for load level D1. The nonlinear analysis provided the initial information on location of the controlled damage zones. The next stage of the study is devoted to detailed analysis of the plastic hinge formation by a local approach.

References

Strength and elastic buckling of a shell of revolution with meridian in the versiera of Agnesi shape

Magdalena Grygorowicz¹, Paweł Jasion², Krzysztof Magnucki³, Piotr Paczos⁴

¹,²,³ Faculty of Mechanical Engineering and Management, Institute of Applied Mechanics, Poznań University of Technology
M. Skłodowskiej Curtis 5, 60-965 Poznań, Poland
e-mail: magdalena.grygorowicz@put.poznan.pl ¹, pawel.jasion@put.poznan.pl ², krzysztof.magnucki@put.poznan.pl ³, piotr.paczos@put.poznan.pl ⁴

Abstract

The paper is focused on a non-classical shape of a shell of revolution with negative and positive Gaussian curvature. The meridian of the shell is a plane curve of the versiera of Agnesi. Geometrical description of the middle surface of the shell is presented. The membrane state of stress is described analytically. The critical pressure and buckling mode are calculated for an example shell with the use of the FEM (the ANSYS system).

Keywords: shell of revolution, versiera of Agnesi, elastic buckling, negative Gaussian curvature

1. Introduction

A shell of revolution is a typical shape of tanks used to storage of liquids or loose materials e.g. cereals. Stress distribution, deformation and stability of such shells is described in literature. The theory of buckling and post-buckling behaviour of elastic structures was described by Budianski [2]. Bushnell [3] described numerical methods in buckling analysis of shells. Krivoshapko [8] or Jasion and Budiansky [2]. Bushnell [3] described numerical methods in buckling behaviour of elastic structures was described by Magnucki [6]. Krużelecki and Trybuła [9] described buckling behaviour of elastic shells of revolution. Elastic buckling of barrelled shell under external pressure was presented by Jasion and Magnucki [6].

The goal of the paper is to analyse strength and elastic buckling of a shell of revolution with meridian in the versiera of Agnesi shape.

2. Geometry of the middle surface of a shell of revolution based on the versiera of Agnesi

Rotation of versiera of Agnesi around the ξ-axis (Fig. 1) forms a surface the most part of which has the negative Gaussian curvature.

Figure 1: Versiera of Agnesi

The versiera of Agnesi curve is defined as

\[ \eta = \frac{1}{1 + \xi^2} = \tilde{f}(\xi), \]  

where: \( \eta = y/a \), \( \xi = x/a \), \( \tilde{f}(\xi) \) - dimensionless radius of the parallel circle, \( a \) - positive constant (the size).

The principal curvature radii are as follows

\[ \tilde{R}_1 = \frac{4\xi^2 + (1 + \xi^2)^{3/2}}{2(1 + \xi^2)^{3/2}}, \quad \tilde{R}_2 = \frac{4\xi^2 + (1 + \xi^2)^{3/2}}{2(1 + \xi^2)^{3/2}}, \]  

where: \( \tilde{R}_1 = R_1 / a \) - dimensionless principal radius of the meridian, \( \tilde{R}_2 = R_2 / a \) - dimensionless principal radius of the parallel circle.

The capacity of the shell

\[ V_0 = 2\pi a^3 I_1, \]  

where \( I_1 = \int_\xi \frac{d\xi}{(1 + \xi^2)^{3/2}} = \frac{1}{2} \left[ \arctan(\xi) + \frac{\eta}{1 + \xi^2} \right]. \]

Assuming the volume of the shell \( V_0 \) and the value of \( \xi_1 \) the constant \( a \) can be determined from Eq. (3).

The mass of the shell

\[ m_1 = \rho_1 A_1 a^2, \]  

where the lateral area

\[ \tilde{A}_1 = 4\pi \xi_1 \tilde{R}_1 d\xi. \]

Knowing the value of the constant \( a \) and assuming the mass of the shell \( m_1 \) the thickness of the shell can be calculated from Eq. (4) for the material of the mass density \( \rho_1 \).

Stresses in the longitudinal and circumferential directions are respectively

\[ \sigma_1 = \tilde{\sigma}_1 \frac{a}{l_1} p_0 \quad \text{and} \quad \sigma_2 = \tilde{\sigma}_2 \frac{a}{l_2} p_0 \]

where dimensional stresses are

\[ \tilde{\sigma}_1 = \frac{1}{2} \tilde{R}_1 \quad \text{and} \quad \tilde{\sigma}_2 = \frac{4\xi_1^2 + (1 + \xi_1^2)^{3/2}}{(1 + \xi_1^2)^{3/2} [4\xi_1^2 + (1 + \xi_1^2)^{3/2}]} \] .
The Huber-Mises-Hencky stresses are

$$\sigma_{eq} = \sqrt{\frac{3}{2}} \sigma \cdot \rho_0,$$  \hspace{1cm} (9)

where

$$\sigma_{eq} = \sqrt{\sigma_{11}^2 - \sigma_{12} \sigma_{21} + \sigma_{22}^2}.$$  \hspace{1cm} (10)

An example of dimensionless stresses distribution based on Eqs 8 and 10 is shown in Fig. 2.

Figure 2: Stresses distribution in the shell

3. Numerical calculation – FEM study

To conduct the FE analysis the mid-surface of the shell has been modelled with the use of shell elements. The model is supported at both ends and in the mid-length of the shell according to Fig. 2.

Figure 2: FE model of the shell

A uniform external pressure of the value of 0.1 MPa has been applied to the whole area of the shell.

The analysis has been made in two steps. The first step was a linear static analysis based on which the deformation and stress distribution have been determined. Second step was a linear buckling analysis as a result of which the buckling shape has been obtained. Parameters used in the analyses: $V_0 = 5 \text{m}^3$, $m_s = 500 \text{kg}$, $\xi = 3$, $a = 1020 \text{mm}$, $t_s = 3.57 \text{mm}$, $p_0 = 0.1 \text{MPa}$, $\rho_s = 7850 \text{kg/m}^3$, $E = 205000 \text{MPa}$, $\nu = 0.3$.

As can be seen in Fig. 3a the shell deforms most in the area of zero Gaussian curvature. It should be noted that in the mid-length the deformation is positive – the circumferential radius increases. The consequence of that is that the equivalent stresses changes rapidly in this region (see Fig. 3b).

The buckling mode is symmetrical according to the mid-plane and has the shape of two longitudinal half-waves and number of circumferential waves.

4. Conclusions

The shell presented in the paper is based on the versiera of Agnesi. It is characterised by a continuous radius of curvature which means that the stress distribution is smooth along the whole meridian. However, since the deformation is not uniform for the whole shell, in the central part the meridional radius increases, the value of stresses changes significantly in this region. This phenomenon is not visible in the solution given by the analytical model in which an uniform deformation due to the membrane state of stress was assumed.

Since the shape of the presented shell is very similar to the pseudosphere analysed by Jasion and Magnucki [7] a similar behaviour of both shells can be expected in the post-buckling range, namely the behaviour can be stable. This will be the subject of further investigations.

Figure 3: Results of FE method analysis: deformation (a), stress distribution (b) and buckling mode (c)

References

New method of modelling nonlinear multi-bolted systems

Rafał Grzejda
Faculty of Mechanical Engineering and Mechatronics, West Pomeranian University of Technology
19 Piastów Ave., 70-310 Szczecin, Poland
e-mail: rafal.grzejda@zut.edu.pl

Abstract

Multi-bolted connections are usually systems of many bodies being in a contact. These systems are composed of such components as fasteners (bolts, washers and nuts), joined elements and contact joints between them. There are known methods for modelling and calculations of parts of typical multi-bolted connections, which are geometrically symmetrical or symmetrically loaded. A novelty is the treatment of multi-bolted connection as a system consisting of components, which can be modelled and calculated as separate subsystems using methods relevant to their properties. The aim of the paper is to present the concept of new modelling method of multi-bolted connections dealt with as multi-bolted systems. The model assumptions and modelling bases of separate subsystems of a physical model of the multi-bolted connection are given. The methods used in the analysis of multi-bolted connection parts are specified. The result of actions described in the paper is the statement of equilibrium equations, which can be applied in calculations both for the assembly condition and the operational state of the multi-bolted system. Calculations results of a selected multi-bolted connection are presented.

Keywords: multi-bolted connection, nonlinear model, equilibrium equation

1. Introduction

Reliability of the designed structure of a machine or device depends primarily on reliability of joints used in this structure. This applies particularly to systems created with bolted connections. Quality of the execution of these connections immensely determines the safety of the machine users. Meanwhile, according to the literature review [7], the comprehensive solution of problems occurring in bolted connections has not been carried out yet. Therefore, the task of modelling bolted connections is still valid and important.

Most of publications dealing with bolted connections concerns modelling single-bolted joints [8]. A second group of papers is related to modelling typical multi-bolted connections as follows:

- beam-to-column connections [2],
- double lap connections [1],
- flange connections [3].

In all these publications, a systematic approach to modelling, calculation and analysis of bolted connections is not undertaken. They concern mainly typical structures with bolted connections.

The subject of studies broached in the paper are geometrically asymmetric and nonlinear multi-bolted connections (Fig. 1). The connections are preloaded and can be externally loaded by an arbitrary force. A physical model of the connection is proposed, which is treated as a system composed of the following subsystems: a pair of flexible joined elements (the flange and the support), a contact layer between joined elements and a set of bolts screwed directly in the support.

The model of the multi-bolted connection is developed based on the model of a joint of a flexible flange element with a rigid support, which modelling and calculations were presented at the 2nd Polish Congress of Mechanics [7]. The new generalized model of the multi-bolted connection is described by the equilibrium equations, which can be used in calculations both for the assembly and operational condition of the connection.

![Figure 1: Multi-bolted connection](image)

2. Structure of the multi-bolted connection

In general, the structure of the multi-bolted connection model arises from the concept presented in Ref. [7]. The model is built with a flexible flange element fastened to a flexible support by means of a set of bolts. A conventional contact layer is introduced between joined elements (Fig. 2).

![Figure 2: Division of the multi-bolted connection into subsystems (1 – subsystem B, 2 – subsystem F, 3 – subsystem C, 4 – subsystem S)](image)

The equation of system equilibrium (Fig. 2) can be written in the form

$$K \cdot q = p$$

(1)

where $K$ is the stiffness matrix, $q$ is the displacements vector and $p$ is the loads vector.
The following four different subsystems can be assigned in the discussed multi-bolted connection:
- subsystem B, built with the bolts,
- subsystem F, which is the flexible flange element,
- subsystem C, related to the conventional contact layer,
- subsystem S, which is the flexible support.

Most often methods applied for modelling components of multi-bolted connections are: the finite element method (FEM), the boundary element method (BEM) and the rigid finite element method (RFEM). Table 1 shows models of the individual parts of subsystems which can be created using FEM.

<table>
<thead>
<tr>
<th>Subsystem</th>
<th>Types of FE-models</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>no-bolt model, plane model, coupled bolt model, rigid body bolt model, spider bolt model, spatial bolt model</td>
</tr>
<tr>
<td>F, S</td>
<td>plane model, spatial model</td>
</tr>
<tr>
<td>C</td>
<td>plane or spatial conventional contact layer</td>
</tr>
</tbody>
</table>

The following four different subsystems can be assigned in the discussed multi-bolted connection:

In view of the introduced division of the multi-bolted system, each subsystem can be separately analyzed with varying degrees of simplification.

3. Numerical results of calculations

Sample calculations were performed for a selected multi-bolted connection, whose model is shown in Fig. 3. In the connection the bolts were replaced by the rigid body bolt models.

The multi-bolted connection was preloaded and then loaded by an external force (Fig. 3). The resulting distribution of operational bolt forces $F_{oi}$ referenced to the preload $F_m$ is illustrated in Fig. 4.

![Figure 3: FEM-based model of the multi-bolted connection](image-url)

![Figure 4: Bolt force values in the connection loaded externally](image-url)

4. Conclusions

The paper presents a general systematic approach to the modelling of arbitrary multi-bolted connections. It can be implemented in both assembly and operational states of the connection. After the adoption of appropriate assumptions, it can also be used for calculations of typical multi-bolted connections.

**References**


An application of the Boundary Element Method to the analysis of initial stability of plates

Michał Guminiak¹, Zdzisław Pawlak²*

¹²Faculty of Civil and Environmental Engineering, Poznań University of Technology
Piotrowo 5, 60-963 Poznań, Poland

e-mails: michal.guminiak@put.poznan.pl ¹; zdzislaw.pawlak@put.poznan.pl ²

Abstract

Initial stability of thin plates is analysed using the fundamental functions and the Boundary Element Method. The proposed approach neglects Kirchhoff forces at the plate corners and equivalent shear forces at the plate boundaries. The governing integral equations take the form of boundary-domain integral equations. The singular and non-singular formulation of the boundary-domain integral equations are presented, involving one and two collocation points associated with a single boundary element of the constant type. A plate domain is divided into rectangular sub-domains associated with suitable collocation points. The plate curvature inside a plate domain is determined by means of double differentiation of basic boundary-domain integral equation or by constructing differential operators in the vicinity of selected internal collocation point.

Keywords: thin plate stability, fundamental solution, Boundary Element Method

1. Introduction

The Boundary Element Method (BEM) in direct singular and non-singular approaches is used in the analysis of plate initial stability. There are well-known works of Bèzine and Gamby [1] and other Authors, who applied the BEM to solve the problems of plate bending. The plate buckling problem using BEM was investigated by Shi [2], Chinnaboon, Chucheepsakul and Katsikadelis [3] and Nerantzaki and Katsikadelis [4]. Guminiak in [5] proposed an alternative to CMM-2015 – 3rd Polish Congress of Mechanics & 21st Computer Methods in Mechanics September 8th – 11th 2015, Gdańsk, Poland

1.2

A plate is compressed only in one direction by the in-plane forces \( N_i \), their distribution is constant along the edge of a single internal sub-domain. The plate curvature \( \kappa_c \) is determined at a selected collocation point '1' (Fig. 1a) of a rectangular sub-surface by means of direct differentiation of the first governing boundary-domain integral equation (1) or by means of a simple difference quotient using deflection of three neighbouring collocation points (Fig. 1b) [5].

\[
\begin{align*}
\lambda & = N_{xx} \quad \text{and} \quad \kappa_c \quad \text{leads to the relation (4), which describes the standard eigenvalue problem, where the eigenvalue} \\
\{A - \lambda I\} \kappa = 0
\end{align*}
\]

Figure 1: Definition of curvature in central collocation point

\[ B = \{G_{nn}, G_{ns}, G_{sn}, -\lambda G_{ns}, 0, 0, 0\} \]

\[ \{A - \lambda I\} \kappa = 0 \]

where \( G_{ij} \) are matrices of suitable boundary and domain integrals, \( B = [B, \phi_k]^T \), \( \varphi \) and \( \kappa \) are vectors of boundary variables, internal support reactions and curvatures, respectively. Moreover, \( \lambda = N_{xx} \), the elements \( \lambda \) are calculated as simple difference quotients of deflections of two neighbouring boundary nodes (\( A \) is the matrix of differential operators). Elimination of vectors \( B \) and \( \varphi \) leads to the relation (4), which describes the standard eigenvalue problem, where the eigenvalue \( \lambda = N_{xx} \).

\[ \{A - \lambda I\} \kappa = 0 \]

where

\[ *This work was supported by internal grant of Institute of Structural Engineering of Poznań University of Technology. \]
The plate curvature can be established directly by a double differentiation of a boundary-domain integral equation (1) or by constructing a difference quotient at the central point '1'.

\[ \kappa = \frac{\Delta^2 w}{(\Delta x)^2} = \frac{w_{x} - 2w_{x} + w_{x}}{(\Delta x)^2} \]  

(7)

Using an equation (1) in its unchanged form.

The set of algebraic equations necessary to compute the eigenvector \( w \) has the form

\[
\begin{align*}
G_{nn} & G_{n} & G_{n} & 0 & B & \lambda G_{nn} \mathbf{K} \\
A & -I & 0 & 0 & 0 & 0 \\
G_{ss} & G_{s} & G_{s} & 0 & q & \lambda G_{ss} \mathbf{K} \\
G_{ss} & G_{s} & G_{s} & 0 & 1 & \lambda G_{ss} \mathbf{K}
\end{align*}
\]

(8)

In the set (8) the first, second and third equations are obtained from the first, second and third equations of (3), respectively. The fourth equation may be obtained by forming the boundary integral equations in order to calculate the plate deflection in the internal collocation points. Hence the investigated displacement vector is

\[
w = \lambda G_{nn} - G_{ss} \tilde{G}_{mn} G_{nn} + \\
G_{ss} G_{nn} \tilde{G}_{mn} \left[ G_{ss} G_{nn} \tilde{G}_{mn} G_{nn} \right]^{-1} \\
\left[ G_{ss} G_{nn} \tilde{G}_{mn} G_{nn} \right] \mathbf{K}
\]

(9)

and

\[ G_{ss} = G_{ss} + G_{ss} \Lambda \]  

(10)

3. Numerical examples

Rectangular plates are analysed in the paper, due to a condition of a supported loading edge. The following designations are assumed: BEM I – singular formulation of boundary-domain integral equations with the vector of curvatures obtained by double differentiation of the boundary-domain integral equation (1) [5]; BEM II – non-singular formulation of boundary-domain integral equations with the vector of curvatures obtained by double differentiation of the boundary-domain integral equation (1) [5]; \( e_1 = 0.01 \) and \( e_2 = \Delta x = 0.01; \) BEM III – non-singular formulation of the first boundary-domain integral equation (1) with the second boundary-domain integral equation obtained for the set of additional collocation points with the same fundamental solution \( w^* \), the vector of curvatures results from difference quotient (7) and fundamental solution \( w^* : e_1 = 0.01, e_2 = 0.1, e_3 = \Delta x = 0.01; \) FEM – regular finite element mesh with number of 200 elements of S4R type. The critical force \( N_c \) is expressed using non-dimensional term \( \tilde{N}_c = N_c / I / |D| \), where \( D \) is the plate stiffness.

3.1. Example 1

A rectangular simply supported plate of dimensions \( l_x = 2.0m, l_y = 1.0m \) and thickness \( h = 0.02m \) is considered. The material properties are: \( E = 205 \) GPa, \( v = 0.3. \) The plate is compressed by \( N_1 \) forces, linearly distributed from zero to its maximum value. Each plate edge is divided into 30 constant-type elements with of same length. The number of internal subsurfaces is equal to 200.

Table 1: Comparison of the results

<table>
<thead>
<tr>
<th>BEM I</th>
<th>BEM II</th>
<th>BEM III</th>
<th>Analytical</th>
</tr>
</thead>
<tbody>
<tr>
<td>( N_c )</td>
<td>155.6620</td>
<td>155.6665</td>
<td>155.6631</td>
</tr>
</tbody>
</table>

3.2. Example 2

A rectangular plate, simply supported on two opposite edges parallel to \( y \)-axis is considered. The plate dimensions are \( l_x = 20.0m, l_y = 10.0m \) and thickness \( h = 0.2m. \) Each plate edge is divided into 40 constant-type elements of the same length. Additionally, the plate rests on 40 internal continuous linear supports. The number of internal sub-surfaces is equal to 200. Material properties are: \( E = 30 \) GPa, \( v = 0.16. \) The plate is compressed by a constant in-plane loading \( N_{c,x}. \)

Table 2: Comparison of the results

<table>
<thead>
<tr>
<th>BEM I</th>
<th>BEM II</th>
<th>BEM III</th>
<th>FEM</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \tilde{N}_c )</td>
<td>19.3800</td>
<td>19.3976</td>
<td>21.3618</td>
</tr>
</tbody>
</table>

4. Concluding remarks

The proposed approach leads to a relatively simple numerical algorithm. A high level of discretization is not required to obtain sufficient accuracy. In addition, the internal sub-surface will be modeled as a four node rectangular element in combination with a finite difference approach necessary to determine the plate curvature. Additionally, the Finite Strip Method algorithms and the corresponding fundamental functions will be also considered in the further analysis.

References


Dynamics of the RUSP linkage mechanism with friction in the joints

Andrzej Harlecki1, Andrzej Urbaś2
1,2 Faculty of Mechanical Engineering, University of Bielsko-Biała
Willowa 2, 43-309 Bielsko-Biała, Poland
e-mail: aharlecki@ath.eu , aurbas@ath.eu

Abstract

Dynamics of a one degree-of-freedom spatial linkage mechanism containing one spherical joint is presented in the paper. It was assumed that friction occurs in the revolute joints and in the prismatic joint of the mechanism, whereas the spherical joint was regarded an ideal one. For the requirements of the method proposed, the mechanism was divided, by a cut joint technique, in the place of the spherical joint into two open-loop kinematic chains, and suitable reaction forces were introduced. Joint coordinates and homogeneous transformation matrices were used to describe the geometry of the system. The equations of motion of the chains were derived using the formalism of Lagrange equations. Cut joint constraints were introduced to complete the equations of motion. In order to determine the values of friction torques in the revolute joints and the friction force in the prismatic joint, in each integration step of the equations of motion, the values of reaction forces and torques in these joints were calculated using the recursive Newton-Euler algorithm formulated for open-loop kinematic chains. The calculations proved the significant influence of friction in the joints on the dynamics of the mechanism considered here.

Keywords: dynamic analysis, linkage mechanism, friction, Stribleck effect

1. Mathematical model of the system

The analyzed one degree-of-freedom RUSP linkage mechanism is presented in Fig. 1. Its links are interconnected and are connected with the fixed base by means of joints – revolute R, universal U (which is a system of two revolute joints R) and prismatic P. The mechanism contains neither passive constraints nor redundant degrees of freedom. The driving link is loaded by driving torque \( t_{(11)}^{(1)} \) and reduced resistance torque \( t_{(11)}^{(2)} \).

The motion of both chains was described by the vectors of joint coordinates \( q^{(1)} \), \( q^{(2)} \), and \( q^{(3)} \).

The equations of motion of both chains were formulated on the basis of Lagrange equations using the algorithm presented in monograph [3]:

\[
\begin{align*}
A^{(1)} & = \left( \begin{array}{ccc}
0 & -D^{(1)} & q^{(1)} \\
0 & D^{(1)} & t_e \\
0 & 0 & \tau_f \\
\end{array} \right),
\end{align*}
\]

where \( n_i \) is the number of links in chain \( i \).

Thus, \( q^{(1)} = \left[ \psi^{(11)} \psi^{(12)} \psi^{(13)} \right]^T \) and \( q^{(2)} = \left[ d^{(21)} \right]^T \).

2. Equations of motion

The motion of both chains was described by the vectors of joint coordinates \( q^{(1)} \), \( q^{(2)} \), and \( q^{(3)} \).

The equations of motion of both chains were formulated on the basis of Lagrange equations using the algorithm presented in monograph [3]:

\[
\begin{align*}
A^{(1)} & = \left( \begin{array}{ccc}
0 & -D^{(1)} & q^{(1)} \\
0 & D^{(1)} & t_e \\
0 & 0 & \tau_f \\
\end{array} \right),
\end{align*}
\]

where \( A^{(c, d)} \) is the number of links in chain \( c \).

The motion of both chains was described by the vectors of joint coordinates \( q^{(1)} \), \( q^{(2)} \), and \( q^{(3)} \).

The equations of motion of both chains were formulated on the basis of Lagrange equations using the algorithm presented in monograph [3]:

\[
\begin{align*}
A^{(1)} & = \left( \begin{array}{ccc}
0 & -D^{(1)} & q^{(1)} \\
0 & D^{(1)} & t_e \\
0 & 0 & \tau_f \\
\end{array} \right),
\end{align*}
\]

where \( A^{(c, d)} \) is the number of links in chain \( c \).

The motion of both chains was described by the vectors of joint coordinates \( q^{(1)} \), \( q^{(2)} \), and \( q^{(3)} \).

The equations of motion of both chains were formulated on the basis of Lagrange equations using the algorithm presented in monograph [3]:

\[
\begin{align*}
A^{(1)} & = \left( \begin{array}{ccc}
0 & -D^{(1)} & q^{(1)} \\
0 & D^{(1)} & t_e \\
0 & 0 & \tau_f \\
\end{array} \right),
\end{align*}
\]

where \( A^{(c, d)} \) is the number of links in chain \( c \).

The motion of both chains was described by the vectors of joint coordinates \( q^{(1)} \), \( q^{(2)} \), and \( q^{(3)} \).

The equations of motion of both chains were formulated on the basis of Lagrange equations using the algorithm presented in monograph [3]:

\[
\begin{align*}
A^{(1)} & = \left( \begin{array}{ccc}
0 & -D^{(1)} & q^{(1)} \\
0 & D^{(1)} & t_e \\
0 & 0 & \tau_f \\
\end{array} \right),
\end{align*}
\]
\[ c^{(3,2)} = J \left[ \sum_{j=1}^{3} T^{(3,2)}_j \dot{q}_j^{(2,1)} \right] \dot{\varphi}_S^{(3,2)} - \left( \sum_{j=1}^{3} T^{(3,2)}_j \dot{q}_j^{(2,1)} \right) \dot{f}_S^{(3,2)} \]  
\[ t^{(2,1)}_j = \left[ t^{(2,1)}_j \ 0 \ 0 \right]^T, \ t^{(1,2)}_j = \left[ t^{(1,2)}_j \ 0 \ 0 \right]^T, \ t^{(1,3)}_j = \left[ t^{(1,3)}_j \ 0 \ 0 \right]^T \]  
\[ s^{(2,1)}_j = \left[ s^{(2,1)}_j \ 0 \ 0 \right] \]  
\[ f_j = \left[ f_{x_j} \ f_{y_j} \ f_{z_j} \right]^T, \ T^{(c,p)}_{c,p} - \text{transformation matrix}, \]  
\[ T^{(c,p)}_{c,p} = \frac{\partial T^{(c,p)}_{c,p}}{\partial q^{(c,p)}} \]  
\[ J = \left[ \begin{array}{ccc} 0 & 0 & 1 \\ 1 & 0 & 0 \\ 0 & 1 & 0 \end{array} \right] = [j_i \ j_j \ j_k], \ \hat{\vec{c}}^{(c,p)}_{c,p} = [c^{(c,p)}_i, c^{(c,p)}_j, c^{(c,p)}_k] \]  

The equations presented here were solved using the Newmark method with the iteration procedure.

In order to determine the values \( t^{(2,1)}_j, t^{(1,2)}_j, t^{(1,3)}_j \) and \( f^{(2,1)}_j \) of friction torques in revolute joints R and the friction force in prismatic joint P, respectively, in each integration step of the equations of motion the values \( \hat{\vec{c}}^{(c,p)}_{c,p} \) and \( \vec{F}^{(c,p)}_{c,p} \) of the reaction forces and torques, respectively, in these joints were calculated by using the recursive Newton-Euler algorithm \[1\].

For the purposes of the method, the models of revolute joints R (Fig. 2a) and prismatic joint P (Fig. 2b) were worked out \[2\].

\[ \dot{\varphi}_S^{(3,2)} = \frac{\partial T^{(c,p)}_{c,p}}{\partial q^{(c,p)}} \]  
\[ \hat{\vec{c}}^{(c,p)}_{c,p} = \left[ \begin{array}{c} \hat{\vec{c}}^{(c,p)}_{c,p} \\ \hat{\vec{c}}^{(c,p)}_{c,p} \\ \hat{\vec{c}}^{(c,p)}_{c,p} \end{array} \right] \]  

and \( \mu^{(c,p)}_{\alpha, \beta, \gamma} \) on these surfaces, respectively (in the case of revolute joints there are three friction surfaces – rotational \( A, B \) and face \( C \)).

It was assumed that the kinetic friction coefficients change along with the change of relative velocity \( q^{(c,p)}_{\alpha, \beta, \gamma} \) in joint \( (c,p) \). The courses of these coefficients (Fig. 3), defined in the paper as kinetic friction characteristics, were determined based on the formula given in article \[5\], as:

\[ \mu^{(c,p)}_{\alpha, \beta, \gamma} = \mu^{(c,p)}_{\alpha, \beta, \gamma} \frac{2 \arctan \left( \frac{q^{(c,p)}_{\alpha, \beta, \gamma}}{1 + q^{(c,p)}_{\alpha, \beta, \gamma}} \right)}{\pi}, \]  

where \( \alpha, \beta, \gamma \) are the assumed parameters.

As can be seen, phases of static friction in the joints were omitted in the analysis, but the Stribeck effect \[4\] in the form of the descending part of the characteristic was taken into account. Such a procedure is allowed because in practice the stiffness of the drive of the linkage mechanisms and the stiffness of their links is sufficiently large, therefore these phases are negligibly short.

Figure 3: Kinetic friction characteristic

The numerical experiments proved the significant influence of friction on the dynamics of the mechanism considered here. They also did not confirm the need to use the constraint stabilization method in order to increase the accuracy of the calculations.

References

Rescaling procedure in the Local Interaction Simulation Approach for shear wave propagation modelling in magnetic resonance elastography

Zahra Hashemiyan1, Pawel Packo2, Wieslaw J. Staszewski3, Tadeusz Uhl4

1,2,3,4 AGH University of Science and Technology, Department of Robotics and Mechatronics
Al. Mickiewicza 30, 30-059 Krakow, Poland
e-mail: zahra@agh.edu.pl 1, pawel.packo@agh.edu.pl 2, w.j.staszewski@agh.edu.pl 3, tuhl@agh.edu.pl 4

Abstract

The Local Interaction Simulation Approach is proposed in the previous study for shear wave propagation modelling Ref. [1]. It indicates the capability of the method for shear wave propagation modelling in Magnetic Resonance Elastography investigations.

This study proposes the upscaling solution method with special material formulation in order to avoid numerical problem and dissipation, especially in wavelength amplitude (modifying the density). The efficiency of the proposed method is checked later by a numerical experiment.

Keywords: upscaling, the Local Interaction Simulation Approach, avoiding the numerical problem in Magnetic Resonance Elastography

1. Introduction

Mechanical properties of soft biological tissues are increasingly required in medical diagnosis to detect various abnormalities, e.g. in liver fibrosis or breast tumor. Since abnormal tissues are often stiffer than the normal one (Ref. [2]). It is well known that mechanical stiffness of human organs can be obtained from organ responses to shear stress waves through Magnetic Resonance Elastography (MRE). Numerical simulations are used in Magnetic Resonance Elastography research studies in order to obtain a forward model that allows for capturing a complex mechanical behavior of various soft tissues (Ref. [3]). In the previous study a three-dimensional model by the Local Interaction Simulation Approach (LISA) analysis was used (Ref. [1]) to model agarose gel phantoms and examine material stiffness and density due to shear wave propagation. Next, the shear wavelength from the simulated LISA model was compared with the relevant result from MRE measurements and analytical models. The simulated results of out of plane displacement components along the radial direction of the phantom models were compared with the experimental MRE results. An example is shown in Fig. 1 for Young’s modulus and density equal $E = 90$ kPa and $\rho = 1.0 \times 1000$ kg/m$^3$, respectively.

Although numerical simulations based on the LISA model were computationally much less expensive than simulations based on the FE model Ref. [1], a significant discrepancy between numerical and analytical results was observed in higher values of elastic moduli and density corresponding to larger wavelengths. Thus a solution method is proposed in order to avoid numerical problems, especially for wavelength amputate problems, by modifying or scaling density in the simulation. The paper aims to improve the model by rescaling wave speed based on introducing scaling density via using the LISA method, later the numerical experiment would be applied to show the efficiency of the method.

Figure 1: Comparison of the simulated FE and LISA displacement waveforms the relevant MRE measurement for the cylindrical agar gel phantom

2. Numerical Problem

In Fig. 1 numerical results from the FE - and LISA - based numerical simulations of shear wave propagation patterns detect almost the same wavelength. Slight differences in these images may be attributed to different formulations of the FE and LISA models and significant numerical damping of the LISA method for the assumed material properties and mesh which affect the wavelength amplitude and cause dissipation. The LISA-based displacement exhibits much larger amplitude values for lower positions than the FE - and MRE - based results. The same displacement for higher positions is out of phase (i.e. appears earlier) when compared with the FE-simulated and MRE-experimental displacements. Discrepancies between numerically (FE and LISA) and analytically (bulk wave propagation solution) estimated results can be observed for higher values of elastic moduli (corresponding to larger wavelengths), particularly for lower densities. The aim is to find solution to overcome this sort of problem, affecting the wavelength amplitude.
3. Solution Method

The LISA – previously used for wave propagation in complex media Ref. [4] – was applied for the MRE shear wave propagation modelling. The method discretises any structure under investigation into a grid of cells. All material properties are assumed constant within each cell but variable between cells. The algorithm is based on a Finite Difference (FD) approximation. Discretisation in time is also used for wave propagation modelling. Recent investigations (Ref. [4]) demonstrate that the new implementation of LISA, based on a parallel algorithm and a Computer Unified Device Architecture (CUDA) available in low-cost graphical cards, offers computational advantage of the method over the FE modelling.

Shear wave propagation in the 3-D cylinder already described in the previous study, was modelled using the LISA approach Ref. [1]. Following these investigations the numerical simulations material properties, boundary conditions and excitation frequencies for numerical simulations were taken from Ref. [1]. The simulated results and experimental shear wave propagation patterns are presented in Fig. 2.

Figure 2: Shear wave propagation patterns for the agarose gel phantom: (a) FE model; (b) LISA model (c) MRE experiment (Chen et al., 2005).

Figure 3 shows the through-thickness cross-section of the wave propagation displacement field for the analysed phantom model. The results, obtained for the 150 Hz excitation frequency, show that the displacement varies across the thickness of the phantom, from a finite value (top) to zero (bottom). This non-uniform displacement distribution indicates that the wave field is strongly affected by the (top and bottom) boundaries; as a result the (global) wavelength is different from the wavelength for the assumed theoretical infinite space case and thus the analysed wave propagation field in the phantom corresponds to the guided wave field.

This idea of scaling density came from the fact that changing density affects the wave speed and wavelength. Thus by scaling and density modification we obtain the desired wavelengths and effect the wave speed.

The paper aims to improve the model by rescaling wave speed based on introducing scaling density using the LISA in order to avoid the wavelength amplitude problem caused by damping. This analysis can serve as an indicator of interfacial conditions for the complex wave propagation in biological tissues.

4. Numerical Experiments:

Numerical experiment was applied to show the efficiency of the method. Following these investigations, first of all the density was scaled then simulated shear wavelengths - calculated for different values of density. Next, the dispersion curve was plotted for every simulation in order to see the influence of density on wave speed. The scaling parameter is then obtained by analysing a plot of dispersion curves, and the wavelength is plotted applying a scaling parameter for every simulation then comparison is done. More details and figures will be presented in a full paper.

References


Numerical study on water hammer with fluid-structure interaction in a straight pipeline fixed with viscoelastic supports

Slawomir Henclik*
The Szewalski Institute of Fluid-Flow Machinery, Polish Academy of Sciences, Fiszera 14, 80-231 Gdansk, Poland
e-mail: shen@imp.gda.pl

Abstract

Water hammer is excited when a pipe flow conditions are disturbed by any reason. The generated pressure wave in the liquid travels along the pipeline and perturbations to functioning of the system may appear. When the pipe is flexible or fixed to the floor with elastic supports, then the pipe motion influences in reverse the flow and the dynamic fluid structure interaction (FSI) occurs. In the paper a straight pipeline fixed to the foundation with viscoelastic supports is assumed. The steady flow is driven by the pressure vessel at the beginning. By means of a sudden closure of the valve at the end the WH is excited. The run of this phenomenon, especially the transient period during the transient, for various stiffness and damping parameters of the supports is numerically examined with the use of an own computer program.

Keywords: pipe flow, water hammer, hydraulic transients, fluid–structure interaction, four equations model, viscoelastic supports

1. Introduction

Water hammer (WH) is excited when pipe flow conditions are disturbed by valves operation, hydraulic machinery load variation, etc. In many cases the pipeline can be assumed rigid and the classic WH theory can be effectively used for modeling of the transient [1,5]. When the pipe is flexible or it is fixed to the foundation with elastic supports then, the interaction between the liquid and the pipe produces the pipe motion, which in reverse, influences the flow. Thus the dynamic fluid-structure interaction (FSI) occurs [4]. Three main FSI factors are mentioned in the literature to couple the liquid flow and the pipe motion. The weakest is the friction between the pipe wall and the liquid. The second mechanism is connected with the Poisson effect and couples the liquid pressures and pipe longitudinal stresses. The third and the strongest one, is a junction coupling (JC) effect. As far as the first two mechanisms are modeled by certain terms in the governing equations, the JC effect is modeled by boundary conditions (BC). JC appears at pipe bends, ends, valves, reductions, etc. and is especially important when the pipe is able to move, which is possible for elastic pipe supports. The JC may produce important behaviour in 2D/3D pipeline system, however a simpler 1D model can also be used to find interesting conclusions. In the current paper the WH-FSI phenomenon in 1D pipeline system fixed to the floor with viscoelastic supports is numerically examined.

2. Numerical model

2.1. Pipeline physical model

A physical model of the pipeline assumed for the analyses consists of a pressure vessel at the beginning, a straight pipe fixed to the foundation with a number of viscoelastic supports (Kelvin-Voigt model is assumed) and a fast closing valve at the end. It is presented in Fig. 1. The WH is excited after closing the valve. The pipe is assumed to be fixed to the pressure vessel with a highly flexible hose. Such a construction allows the pipe to vibrate as a whole body during the transient.

2.2. Mathematical model and numerical method

The 1D four equations (4E) model of WH-FSI is used at the current study. This model is governed by four hyperbolic partial differential equations (PDE) of the first order, two for 1D liquid flow and the other two for the longitudinal motion of the pipe [2,4]. These equations govern two coupled, elastic waves – the WH and the precursor (PC) ones, and allow to find the liquid pressure \( p \), liquid velocity \( v \), pipe cross-section velocity \( w \), pipe stresses \( \sigma \) as a function of position \( x \) (along the pipeline) and time \( t \). To find the numerical solutions the method of characteristics is used. The governing equations are transformed to the compatibility ones (CE) which are integrated within the properly selected time step to get finite difference equations. The solutions to the latter are found marching in time on the properly designed space-time grid. The adjustment of WH and PC wave-speeds was used for numerical application (the wave-speeds have been approximated to form a ratio given with small integers). This model was applied to the authors, computer program used for the current numerical analysis.

*The results presented in the paper have been partially developed within the research project No. N N504 478839 sponsored by the Ministry of Science and Higher Education of Poland
2.3. Boundary conditions

The important part of the model described above are the BC. Beside the standard hydraulic BC at the valve [3] and at the beginning of the pipeline the essence of the current problem is the BC at a junction where the viscoelastic support is fixed to. This BC is formulated as a differential equation of motion (EOM) of the junction. If the junction displacement is \( z(t) \) and its mass is \( m \), the EOM has the following form (\( k \) is the stiffness and \( b \) - damping coefficient of the spring):

\[
mz + bz + kz = Q + F - (Q_0 + F_0)
\]  

(1)

On the right-hand side of the EOM there are pipe forces \( Q \) and liquid ones \( F \) taken relative to their steady state values. Using the difference form of the CE to the right-hand side of the EOM and the Newmark method to the left one, the solution \( z(t) \) for subsequent time moment was found, to be used in the numerical scheme. The computed junction motion allows to estimate the energy dissipated at the supports.

3. Results and conclusions

Numerical results were computed for a straight pipe fixed with six, same, equally spaced viscoelastic supports. To focus on the JC effect connected with the supports and the closed pipe end, the Poisson effect and the friction between the liquid and the pipe-wall are neglected. The valve is closed instantaneously.

In Fig. 2 the pressure records for highly elastic support (\( \kappa = 0.1 \)) and various damping coefficients are presented.

Figure 2: Pressure record at the valve for \( \kappa = 0.1 \)

The energy dissipated at the supports in time for these cases is plotted in Fig. 3.

Figure 3: Energy dissipated at the supports for \( \kappa = 0.1 \)

The relative stiffness \( \kappa \) and damping ratio \( \xi \) of the support system are determined with the following formulas

\[
\kappa = \frac{k_{tot}}{4\pi f_{WH}^2 m_{tot}}
\]  

(2)

\[
\xi = \frac{b_{tot}}{2\sqrt{k_{tot}m_{tot}}}
\]  

(3)

In the above equations \( k_{tot} \) is the sum of stiffness coefficients of all the supports, \( b_{tot} \) is the total damping coefficient and \( m_{tot} \) is the mass of the pipe with the water inside. The classic WH frequency is given in \( (c = WH \text{ wave celerity, } L = \text{ pipe length}) \)

\[f_{WH} = \frac{c}{4L}\]

(4)

The results in Fig. 2 are compared to the classical WH (rigid piping) record (dashed line). In Fig. 4 the pressure runs for much more rigid pipe fixing (\( \kappa = 10 \)) are presented.

Figure 4: Pressure record at the valve for \( \kappa = 10 \)

The numerical analysis results, allowed the author to formulate preliminary conclusions of the system behaviour.

- Viscoelastic supports for the whole pipe allow for its motion and JC effect due to FSI at the pipe close end. It is stronger for lower stiffness of the supports.
- The wave-speeds ratio of PC and WH waves, equal 50:13 (\( \approx 4:1 \) ) can be observed at the pressure records.
- In general, pressure reductions (relative to classic WH) are observed. A temporary pressure increase may also happen.
- This reduction is supposed to be mainly the effect of energy dissipated at the supports and is of greater importance for lower supports stiffness (pipe motion is more significant).

References

Numerical models for live load distribution in a multi-span steel-concrete composite bridge

Janusz Holowaty¹, Gabor Zimny²

¹Faculty of Civil Engineering and Architecture, West Pomeranian University of Technology
Piastów 50, 70-311 Szczecin, Poland
e-mail: janusz.holowaty@zut.edu.pl

²PPDM
Wilków Morskich 6/9, 71-063 Szczecin, Poland
e-mail: gaborz@vp.pl

Abstract

Numerical models developed in the analysis of road steel-concrete composite bridge decks are presented. For a comparison of the models, the results of load test measurements on two parallel composite decks of multi-beam structures along a newly-constructed dual express road are used. A grillage model was developed in the design and in the verification of load test deflections, resulting in a good compliance. In the paper, a more sophisticated 3-D numerical model is also presented as expected to bring more precise results. The numerical models produce satisfactory results for the lateral live load distribution in composite decks. Good compliance with field load test results is attained. High load distribution performance in the composite bridge decks is ensured.

Keywords: composite bridge, composite action, proof load test, numerical model

1. Introduction

Following the construction of a bridge, a proof load test is usually carried out in order to analyse the bridge behaviour and the live load distribution. The basic scope of proof load tests is the evaluation of the bridge design assumptions and to discover any inconsistencies in the bridge works. Many parameters influence the behaviour of a composite bridge deck. In the design of a steel-concrete composite bridge, grillage models are generally used as the most comprehensive and well-understood. A grillage model is also used to verify the load test results for the bridge structure presented. The calculated deflections compare with the measured values well. However, a more advanced model is used for accuracy. As the bridge superstructures for a dual express road are multi-beam decks, the live-load distribution effect onto individual composite girders is given the greatest attention.

2. Bridge decks and load tests

The bridge was opened to traffic in 2010 as the last structure on a new dual carriageway expressway S3 between Szczecin and Gorzów Wlkp., northwestern Poland. The total length of the bridge is 164.4 m. It comprises two five-span composite decks of the same straight structure for each carriageway (Fig. 1). The bridge is located in swampy terrain and covers a protected peat bog (Fig. 2). The cross section of one deck comprises six built-up steel girders spaced at 2.0 m and a reinforced concrete (RC) slab. A slab depth of 0.21 m was chosen.

The decks are continuous five-span structures, with span lengths of 28+3×35+28 m. The total deck width excluding the edge beams is 12.2 m. The steel grids consist of 1.31 m deep constant-depth longitudinal steel plate girders. The girders are braced transversely by steel cross beams of a built-up structure. Structural non-alloy steel of strength grade S355J2 is used in the steelwork and concrete grade C32/40 in the slabs. In the design and the load test calculations, 2-D models (plane grillage) were used as grillage analogy methods, generally to produce more accurate results than the simplified bridge deck analysis methods, which are sometimes too conservative.

Proof load tests, both static and dynamic, were performed to satisfy code requirements. Six lorries of 27 t gross weight were used in the static tests. The deflections were measured on all the longitudinal girders. This enabled deflections over the whole cross sections of the decks under the same load to be mirrored. The proof load schemes were mainly asymmetrical in the cross sections of the decks for verifying the external girders at midspan as the critical elements in the design. For the purpose of load distribution verification, two lorries (2×27 = 54 t) were placed transversally in the sections measured. One span in each deck was tested for this loading. The results of the deflections measured in interior span of the deck to Szczecin are shown in Fig. 3. The deflections measured under the load tests are compared with the numerically-computed deflections (2-D and 3-D models). The deflections measured are highly satisfactory.
in the magnitude and in their distribution for grillage model. The 3-D model reflects more transverse rigidity of the deck.

The grillage model for load test verification, similar to the design model, assumes that the concrete deck slab is uncracked over the internal supports and there is full composite action between the steel longitudinal girders and the concrete slab. The same is followed in the 3-D model. This means that global elastic analysis assumptions are met in the calculations.

### 3. Live load distribution

In composite decks, the longitudinal girders are the most critical members as they transmit loads to the supports. In bridge design, longitudinal girders should carry approximately the same loads after distribution through the deck slab. In order to compare the numerical models, the live load distribution technique is applied, using influence lines of distribution coefficients $\kappa$ calculated on deflection distribution.

The results of load distribution in the forms of influence lines for the models under comparison are shown for the external and internal girders in Fig. 4. As expected, the differences are small as the response of the girders is determined by flexural deformations. The 3-D model gives a more precise estimation as the shell elements describe the deck slab stiffness better. The grillage model may be improved by adding additional ingredients to the overall torsional stiffness of the nominal girder members. The minor ingredients are usually neglected in the technical computation of bridge decks. The high value of distribution coefficients for the external girder needs special consideration in the technical design.

### 4. Conclusions

Proof load tests and numerical assessment of multi-span bridge superstructures under test loading reflect the general performance of the steel-concrete composite decks. A comparison of calculated and measured deflections is quite good. The grillage model is an approximation of a real composite bridge deck and even when used correctly, there may be inaccuracies. A 3-D numerical model is rather more precise; however, interpreting the results may be tedious. The models achieve similitude between the numerical nominal elements and the bridge deck sections. In both models, assumptions of linear and elastic behaviour in the materials are followed. Full composite action between the steel girders and the deck slab is assumed. The elastic behaviour of the structures is confirmed by proof test loading.

For comparison of the numerical models, the live load distribution technique is adopted in its classical form. A good agreement with the field test results show that for the analysis of composite slab-on-girder decks for this range of span length, a plane grid is a reasonable and effective numerical model. A more advanced three dimensional model confirmed the usage of a simple but nevertheless accurate grillage model.
FE analysis on buckling of cylindrical silos composed of horizontally corrugated sheets

Piotr Iwicki¹, Michał Wojciech², Mateusz Sondej³, Karol Rejowski³, Natalia Kuczyńska⁴, Jacek Tejchman⁵

¹,2,3,4,5,6 Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland

e-mail: piwicki@pg.gda.pl ¹, mwojciech@pg.gda.pl ², matsonde@pg.gda.pl ³, rejowskikarol@wp.pl ⁴, kuczynska.nat@gmail.com ⁵, tejchmk@pg.gda.pl ⁶

Abstract

The paper deals with the stability of cylindrical metal silos composed of horizontally corrugated sheets with and without vertical stiffeners. Three different variants of analysis were carried out using a commercial FE package ABAQUS [1]: linear buckling, dynamics with geometric non-linearity and dynamics with both geometric and material non-linearity. The FE analysis was performed with or without a bulk solid. In the second case, axisymmetric and non-axisymmetric loads imposed by a bulk solid following Eurocode 1 were considered. The calculated buckling forces were compared with the permissible ones of Eurocode 3.

Keywords: silo buckling, bulk solid, hypoplasticity, FE analysis, cylindrical shell, corrugated wall, Eurocode approach

1. Introduction

Thin-walled metal cylindrical shells are frequently used in the silo industry. They are vulnerable to buckling caused by the wall friction force caused by the interaction between the silo fill and the silo wall, particularly during eccentric discharge, which is usually difficult to avoid with regard to a non-homogeneous character of bulk solids. The buckling strength of shells depends on many different factors such as: form and amplitude of initial geometric imperfections, loading and material imperfections, type of joints, boundary conditions at ends, flow pattern, level of internal pressurization, wall roughness and stiffness of the stored bulk solid. The buckling strength of silo shells containing bulk solids at rest may be significantly enhanced while compared to empty silos due to both internal pressure and lateral support produced by the silo fill. On one hand, the internal pressure in bulk solids acting on the silo wall straightens wall imperfections and increases the buckling strength. On the other hand, the silo walls are supported by the fill which restrains them against buckling.

The paper deals with the stability of thin-walled cylindrical metal silos composed of horizontally corrugated sheets. Two different silo structures were analysed with or without vertical stiffeners. The linear buckling and dynamic nonlinear FE analysis was carried out [1]. The calculations were performed with and without the bulk solid. In the second case, axisymmetric and non-axisymmetric loads imposed by a bulk solid following Eurocode 1 [2] were taken into account. The calculated buckling forces were compared with the permissible ones given by Eurocode 3 [3]. The buckling capacity according Eurocode 3 [3] uses two alternative methods which depends upon the stiffeners distance d_s. For d_s<d_s,max the silo wall is treated as the equivalent orthotropic shell (method 'A') and for d_s>d_s,max the stiffener resting on the elastic foundation provided by the silo wall may be taken into account (method 'B'). The silo buckling resistance calculated with the method 'A' was 3.8 times higher than required (λ=3.8) but with the method 'B' (which should be used for this silo with respect to d_s,λ), the buckling load bearing capacity was not sufficient (λ=0.4).

2. FE input data

2.1. Silo composed of corrugated sheets with vertical stiffeners

Numerical calculations were carried out for the existing silo with height h_s=17.62 m and diameter D_s=8.02 m. The silo was made from corrugated sheets and strengthened by uniformly distributed 18 vertical stiffeners (stiffener spacing d_s=1.4 m). The 2.5 m-long open thin-walled profiles were used for columns. The fully integrated 4-node quadrilateral S4 shell elements (size: 0.25 m × 0.25 m) and beam B33 were used in the model 'A'. The element size in the model "B" was 0.01 m × 0.09 m for the corrugated wall and 0.01 m × 0.03 m for the columns. A total number of 546 elements was 885’000. The silo wall loads induced by maize were calculated according to Eurocode 1 [1]. During axisymmetric emptying, the standard maximum wall normal and shear stress in the bin were p_w=30.52 kPa and p_w=19.38 kPa, and during non-axisymmetric emptying, they increased up to p_w=36.26 kPa and p_w=26.68 kPa [2]. In numerical analyses the load factor λ was always related to the wall shear stress of p_w=26.68 kPa.

Linear buckling analysis was conducted for a simplified 3D FE model 'A' (with the equivalent orthotropic shell and beam elements) and a full shell model 'B' were the corrugation and profiles were described with shell S4 elements. Linear buckling analysis was performed for the model 'A' considering two load cases: 1) shear traction ('S') and 2) shear traction and normal pressure ('S+N'). The dynamic non-linear analyses (GNA and GMNA) were carried out for the model 'B' with the 'S+N' load.

2.2. Silo composed of corrugated sheets without vertical stiffeners

The medium-size silo was H_s=8.4 m high, its diameter was D_s=2.67 m and wall thickness t=1 mm. The 4-node thin shell elements S4R with a reduced integration point were employed to represent the silos walls and 8-node linear brick elements C3D8R to depict the bulk solid [1]. The total number of the finite elements was 287’000 (45’000 - silo wall and 242’000 - bulk solid). The steel was assumed elastic-perfectly plastic. The behaviour of bulk solids (sand and wheat) was described with a hypoplastic constitutive model, which took into account the salient properties of granular materials [4].

*We acknowledge the support from Grant WND-POIG.01.03.01-00-099/12-01 financed by Polish National Centre for Research and Development NCIR (2013-2015).
3. Results of stability analyses

For the silo of Section 2.1, the buckling load factor $\lambda$ was $\lambda=2.47$ for the 'S' load (Fig. 1a) and 3-times higher for the 'S+N' load $\lambda=7.35$ (Fig. 1b). Compared to Eurocode 3, the silo had an insufficient buckling resistance. The dynamic limit load factor was $\lambda=3.22$ (GNA) and $\lambda=1.45$ (GMNA). The failure deformation mechanism was characterized by a local stability loss of silo column walls (GNA) (Fig. 2a) or by reaching the yield stress at the wall connectors between two different column profiles (GMNA) (Fig. 2b).

![Figure 1: Simplified silo model: linear buckling analysis (LBA) for two load cases: a) 'S' and b) 'S+N'.](image1)

![Figure 2: Full shell model of silo: deformation for limit load factor: a) GNA and b) GMNA.](image2)

![Figure 3: Distribution of normal wall pressure $p_{wf}$ and vertical wall traction $p_{wf}$ along silo height $H$ (dashed line – Eurocode 1 [2] and continuous line - FE result).](image3)

![Figure 4: Evolution of buckling load factor $\lambda$ and vertical reaction wall force RF versus vertical displacement $u$ of top of silo with sand (dashed line – Eurocode 1 [2] and continuous line - FE result).](image4)

4. Conclusions

The Eurocode approach [3] for the buckling strength is too conservative, in particular for the large stiffener spacing ($d_s>d_{s,max}$). The FE analyses show that the global elastic buckling does not occur. The failure mechanism is mainly related to the local stability loss or to local plastification.

The calculations results of silo containing sand and wheat clearly demonstrate that the bulk solid significantly increases the buckling strength of silos during filling as compared to the silos loaded according to Eurocode 1 [2]. The strengthening effect of the solid is 2000% for sand and 450% for wheat, for the wall thickness of 1 mm. The buckling strength increases with increasing material stiffness.

References


Numerical methods for the assessment of bridge safety barriers

Kazimierz Jamroz¹, Stanisław Burzyński², Wojciech Witkowski³, Krzysztof Wilde⁴, Grzegorz Bagiński⁵

¹ Highway Engineering Department, Faculty of Civil and Environmental Engineering, Gdansk University of Technology Narutowicza 11/12, 80-233 Gdańsk, Poland  
e-mail: kjamroz@pg.gda.pl

²,³,⁴ Department of Mechanics, Faculty of Civil and Environmental Engineering, Gdansk University of Technology Narutowicza 11/12, 80-233 Gdańsk, Poland  
e-mail: staburzy@pg.gda.pl³, wojwit@pg.gda.pl ³, krzysztof.wilde@pg.gda.pl ⁴

⁵ SafeRoad RRS Polska Sp. z o.o.  
Leszczynowa 6, 80-173 Gdańsk, Poland  
e-mail: grzegorz.baginski@saferoad.pl

Abstract

The paper presents a numerical study of crash tests for bridge safety barriers. The analysis is conducted with the use of an explicit code LS-DYNA. The simulation procedures are verified on the benchmark problem of the car hitting a rigid obstacle. The numerical analysis is conducted for vehicles crashing into the bridge barriers of different types on concrete and steel bridges.

Keywords: crash tests, FEM analysis, bridge safety barriers

1. Introduction

Road accidents continue to tragically affect many families in Poland. Many of the crashes are run-off-road accidents whose consequences include rollovers or hitting roadside objects. Despite significant progress in hazard reduction over the last decade, these types of accidents have claimed more than 6 300 lives on Polish roads [1]. Road and bridge safety barriers, crash cushions and passive support structures are road safety devices that reduce the probability of a run-off-road accident and its consequences. With high traffic volumes and high vehicle speeds, national roads are most frequently provided with road safety devices. National roads account for about 19 000 km of roads and have 6 900 bridges of the total length of 367 km. This represents almost 2% of national roads. While bridges are the scene of about 0.5% of all accidents, the severity is much worse than on other road sections.

The operation and performance of safety barriers and supports is set out in the standards which are systematically improved and published. Standard PN-EN 1317 defines the requirements for “road limiting systems” and tests that should be conducted before the structures are considered fit for purpose [2]. The standards describe classes of safety barrier performance. Barriers are characterised based on their functional features (containment levels, barrier deformation and impact of dynamic forces on car occupants) that are achieved in crash tests. Safety barrier performance classes as defined in the standard, depend on the speed, weight and angle impact as confirmed in crash tests. The technical parameters of barriers are defined in road authority requirements. In Poland in 2010 the General Directorate for National Roads and Motorways issued guidelines for using safety barriers on the national roads network [3]. In 2014, the safety barrier guidelines were to be updated, but after numerous stakeholder discussions the document was not implemented because many of the proposals did not seem justified.

2. Bridge safety barriers

Bridge safety barriers and the way they should be used are not regulated in design and construction practice in Poland.

A number of publications address topics such as experimental tests, modelling, simulation, validation and experimental verification of road crash tests [4], [5]. In particular, bridge safety barriers were studied by Barnas and Edl [6] on the performance of concrete barriers upon heavy impact, Thai [7] on the performance of steel and composite bridge safety barriers and Atahan and Cansiz [8] on the performance of different road and bridge transition structures. There is no research, however, on the distribution of forces in anchors that fix bridge safety barriers and the reinforcement of the bridge structural elements.

The basic problems that are still unresolved include: selecting the right barrier containment class for the level of risk and modal split, the effects of additional elements of bridge barriers and curb height on changes in functionality, linking rigid bridge barriers with deformable safety barriers and defining the strength of forces on bridge support structures for the different containment levels and types of engineering structure design (steel vs. concrete).

3. FEM analysis of bridge safety barriers

Physically a crash is a phenomenon of a dynamic violent nature, lasting, as a rule, not more than 1s. However, during this short time, large deformations occur that are accompanied usually with extensive damage. Usually, the materials exhibit rate dependence. From the FEM modelling viewpoint, to account for these observable effects it is necessary to apply FEM codes that use explicit time integration schemes to compute the evolution of internal variables. Since these schemes are conditionally stable, it is very important to apply extremely short time increments for the integration of equations of motion. This necessitates the use of high-end computers (or computer clusters) to run the calculations. Therefore, substantial
experience in FEM and structural dynamics is required to properly interpret the results. One of the most popular explicit codes, targeted at crash test simulations is LS-DYNA. In order to verify the application of the code and the numerical effort associated with the simulations, the standard problem, i.e. analysis of a car (Toyota Yaris 2010) which hits a rigid wall, is calculated and discussed.

The Toyota Yaris model has been developed by the National Crash Analysis Center (NCAC) of the George Washington University under a contract with the FHWA and NHTSA of the US DOT. The model consists of 998218 nodes and 974383 finite elements of various types. All material data were collected in laboratory tests, with specimens cut from actual car components. The selected simulation time is 0.15s. The calculations were carried out at the Academic Computer Center in Gdańsk. The total time of simulation on a workstation (8-core Xeon processor@3.4 GHz, 16 GB RAM) amounted to 19 hours and 39 minutes. Figures 1 and 2 show the initial and final configuration of the benchmark crash test. The results of the analysis show very good consistency with full-scale crash test data obtained experimentally [9].

The next stage of the research is the problem of a car hitting concrete road barriers on bridges (Fig. 3) with dynamics of bridge structural elements taken into account. The impact force leads not only to damage of the barrier, but also affects bridge secondary elements like steel reinforcement. That may lead to the barrier slipping from the bridge, even though the barrier remains functional.

Figure 1: Initial configuration of the benchmark crash test

Figure 2: Final configuration of the benchmark crash test (time = 0.15s)

Further research is going be conducted on crash tests for various bridge safety barriers for different bridge structures. Two types of bridges are considered:
- concrete bridges with two types of barriers: concrete and steel (standard solutions and new types),
- steel bridge structure with a steel barrier fixed directly to the steel orthotropic bridge deck.

References

[9] http://www.ncac.gwu.edu/vml/archive/ncac/vehicle/yaris-v1m.pdf (access: 2.06.2015)
Abstract

The paper presents a design of switching tracking controllers for a system of one of actuators failed during motion. The controllers are model based and the system is represented by 3 degrees of freedom manipulator which may work in horizontal or vertical planes. The control goal is to enable the end effector of a broken manipulator completing tracking a predefined task as good as possible and then get back to its rest position. The novelty of the presented research, comparing to the results reported in the literature, relies upon the control goal formulation and the subsequent controller design. Specifically, we require the manipulator to complete the task not only to return to the rest position. Simulation studies confirm good performance of the proposed switching controllers turned on after the actuator failure.

Keywords: model based tracking, underactuated systems, nonlinear controllers

1. Introduction

The paper presents control of constrained mechanical systems, where constraints may origin from underactuation and tasks required to be performed. Systems are referred to as underactuated due to the number of control inputs which is less than the number of its degrees of freedom. They may be designed underactuated to reduce their weight, failure rate, make them cheaper, and simpler in operation. Examples of these are manipulators and spacecraft, surface and underwater vehicles [1-4]. A problem of a system control after failure of one or more of its actuators is important and even more challenging from the practical view point. An actuator failure may cause damages of the system or its work environment, a task is not completed, and its repetition may be hard, expensive or even not possible.

Underactuated control systems are often referred to as second order nonholonomic. This is due to motion equations of unactuated degrees of freedom, which are treated as constraint equations, which are generally not integrable [5]. Control of nonholonomic systems requires special control methods due to the Brockett result [6]. Most research reports developments of sophisticated control strategies for systems underactuated by their design [7]. Control strategies for underactuated systems when other constraints are put on were developed in, e.g. [8].

From the practical point of view, a more interesting problem is to design a switching controller, which can be turned on when an actuator failure occurs. The motivation is then to design and put into operation a controller to which a regular one can be switched to for completing task tracking.

The paper presents a development of such a switching tracking controller which can be turned on when an actuator fails during a system motion.

The novelty of the presented research, comparing to the results reported in the literature, relies upon the control goal formulation and the subsequent controller design. Specifically, we require the manipulator to complete the task as good as possible but not only to return to the rest position. Numerical studies illustrate the performance of the emergency tracking controllers applied to the manipulator.

2. Tracking control of a fully actuated 3-dof manipulator

Consider the 3 degrees of freedom manipulator, whose geometric and inertia properties are gathered in Table 1.

<table>
<thead>
<tr>
<th>Manipulator</th>
<th>Link 1</th>
<th>Link 2</th>
<th>Link 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass [kg]</td>
<td>m₁ = 0.6</td>
<td>m₂ = 0.6</td>
<td>m₃ = 0.28</td>
</tr>
<tr>
<td>Central moments of inertia [kgm²]</td>
<td>J₁ = 0.0039</td>
<td>J₂ = 0.0039</td>
<td>J₃ = 0.00087</td>
</tr>
<tr>
<td>Link length [m]</td>
<td>L₁ = 0.22</td>
<td>L₂ = 0.22</td>
<td>L₃ = 0.06</td>
</tr>
<tr>
<td>Distance between a mass center and a rotation axes [m]</td>
<td>l₁ = 0.054</td>
<td>l₂ = 0.05</td>
<td>l₃ = 0.018</td>
</tr>
</tbody>
</table>

The manipulator control dynamics is gathered by the equations

\[ M(\phi)\ddot{\phi} + C(\phi, \dot{\phi}) + D(\phi) = \tau \]  

where \( \phi = [\phi_1, \phi_2, \phi_3]^T \) - state vector, \( M \) - inertia matrix, \( C \) represents centrifugal forces, friction and external disturbances, \( D \) - gravity dependent term, and \( \tau \) - control torque vector. External disturbances are added as the Matlab functions

\[ M_d = 0.006\operatorname{rand}(i)\sin t, \quad i = 1.2.3 \]

Friction is added in the joints according to the Tustin model [9].

The control goal is to track a desired trajectory by the manipulator end effector. This reference trajectory is presented in Fig. 1a. It is a curve that requires the end effector multiple velocity and its direction changes.

A control torque, after a dynamic feedback linearization of the dynamic control model (1), can be applied, i.e.

\[ \tau = M(\phi)u + C(\phi, \dot{\phi}) + D(\phi), \]

where \( \alpha \) is a new control input which is a PD with correction

\[ u = \ddot{\phi} + k_1(\dot{\phi} - \phi) + k_2(\phi - \dot{\phi}) \]

Tracking results are presented in Fig. 1b. In what follows, the 3 link planar manipulator model is referred to as 3LP and the 3 link vertical one – 3LV.
3. Switching tracking controller design

An n-link planar underactuated manipulator is completely controllable if and only if the first joint is actuated [10]. For this reason we assume the failure of the third actuator after 8 seconds of motion tracking.

Two switching controllers are designed. One dedicated to the 3LP – switching I (SI) and one dedicated to the 3LV – switching II (SII). The idea of the two control methods is based on iterative prediction of motion of an unactuated link. The prediction uses parameters from the previous motion step and adjusts them to the reference trajectory in a desired time.

The SI design for a broken 3LP is as follows:
1. Predict motion at \( t_{n-1} \), based upon the angular position \( \varphi_3 \) and velocity \( \dot{\varphi}_3 \) of an unactuated joint at \( t_n \).
2. The angular acceleration \( \ddot{\varphi}_3 \) is determined from dynamic equations assuming that the system motion is free (actuators are turned off).
3. In SI after \( t_{n-1} = t_{n-1} \), an angular position can be predicted assuming that
   \[
   \varphi_{n+1} = \varphi_n + \ddot{\varphi}_3 \Delta t + \frac{\ddot{\varphi}_3^{\text{free}} (\Delta t)^2}{2}.
   \]
   The SI is also based upon the PD controller. The SI does not work well for the 3LV, since additional compensation is needed due to the gravity force. An SII is needed.

The SII design for a 3LV is as follows:
1. Predict motion at \( t_{n-1} \), based upon the angular position \( \varphi_3 \) and velocity of an unactuated joint at \( t_n \).
2. The angular acceleration \( \ddot{\varphi}_3 \) is determined from dynamic equations assuming that the actuators between \( t_0 \) and \( t_{n-1} \) are on and do not change their magnitudes. This is due to gravity that has to be compensated.
3. In SII after \( t_{n-1} = t_{n-1} \), an angular position can be predicted assuming that
   \[
   \varphi_{n+1} = \varphi_n + \ddot{\varphi}_3 \Delta t + \frac{\ddot{\varphi}^{\text{free}}_3 (\Delta t)^2}{2}.
   \]
   Comparing to SI, a term \( \ddot{\varphi}^{\text{free}}_3 \) shows up in the SII design.

4. Switching tracking - a simulation study

Simulation results of the reference trajectory emergency tracking with EMECI for 3LP manipulator model are presented in Figs. 2a and 2b. In switching tracking by SI for the 3LP, the unactuated joint is sensitive for disturbances and motion direction changes. It starts oscillating and so does actuated joints. Control torques for SI in actuated joints get about 2 times larger than for tracking an actuated manipulator. In figure 4 tracking lasts for 68sec. An arrow shows the point at which the unactuated link starts rotating about its joint.

5. Conclusions

Both, SI and SII were tested for manipulators of different link numbers, lengths and masses. The tracking performance is satisfactory and the desired task can be continued for some time and then the manipulator may be brought to its rest position. Experiments would be anticipated, since real motors may not allow for control moment magnitudes changes as predicted by simulations. Optimization or additional controller corrections are required to improve the switching tracking performance.

References

On calculation of effective material properties using RVE method by parallelized FE code for shell applications

Paweł Jarzębski¹, Krzysztof Wiśniewski²∗
¹,²IFTR, Polish Academy of Sciences
Pawińskiego 5B, 02-106 Warsaw, Poland
e-mail: pjarz@ippt.pan.pl ¹, kwisn@ippt.pan.pl ²

Abstract

The paper concerns parallelization of an FE code for machines with shared memory in order to speed up computations of large models. The loop was parallelized over elements in the research code FEAP using OpenMP, which required several modifications of the code and a specific method of synchronization for assembling, see [2] for details. The parallel solver was also applied.

Performance of the parallelized FEAP, designated as ‘ompFEAP’ is demonstrated in calculations of effective properties of materials using the RVE method. Two RVE examples are computed, for a heterogeneous metal-ceramic composite and for a ceramic foam with a complicated micro-structure. We conclude that ompFEAP provides a very good speedup and efficiency causing only a small increase in memory usage.

Keywords: parallelization, OpenMP, finite element method, FEAP, RVE, shells

1. Introduction

The Representative Volume Element (RVE) method is a computational technique developed to determine average properties of composite, porous and cellular materials. In the two-scale approach, these properties can be subsequently used in computations of shells [3], and the solid-shell elements are particularly well suited for this purpose. In the RVE computations, typically, 3D finite elements are used, models have the size of millions of degrees of freedom (see [6]) and must be repeated many times while building a database of material properties.

Concurrent capabilities of multi-core processors and multimachine clusters should be used, requiring advanced computational techniques. The minimum is to use a FE code with an available parallel solver, such as PARDISO, MUMPS, PaStiX, HSL and others, relatively easy to implement. The next step is parallelization of computations of the FE matrices and vectors, e.g. using OpenMP. In the third step, the so-called hybrid approach, with the domain decomposition is performed, e.g. by METIS, computations for sub-domains are scattered over a cluster of computers, which requires, e.g. MPI. Implementation of these techniques in a large and complicated existing serial FE code requires a significant programming effort.

In the paper a parallelization of the research code FEAP [1] is considered using the OpenMP library to make computations on a single machine (not a cluster) more effective. The parallelized code is designated as ‘ompFEAP’.

Efficiency of parallelization is assessed by computing average properties of two advanced materials: (1) a metal-ceramic composite and (2) a ceramic foam. It is shown that the applied parallelization significantly reduces times of execution and causes only a small increase in memory usage.

2. Parallelization of FEAP

The FEAP [1] is a research FE code used at many universities, developed by Prof. R. L. Taylor at the UCB. A serial and a cluster (MPI) versions of FEAP exist, the version for one shared memory machine with multi-core processors does not, so the task was undertaken developing it, see [2]. The parallelization of FEAP involves: (1) implementing a parallel loop over elements, and (2) adding a parallel solver.

2.1. Parallelization of the loop over element

The main difficulty in parallelization of the loop over elements is caused by an existing architecture of this code. Several modifications in FEAP were required, see [2] for details. The main features of the implementation are as follows:

1. assembling of elemental matrices is performed immediately after generating, without an additional storage,

2. only standard directives of OpenMP are used. The crucial part of the loop is the assembling of elemental matrices into a global matrix, in some papers designated as the ‘reduction’, as there is the possibility of the race condition. All the available directives for the mutual exclusion synchronization of OpenMP were implemented and tested; the directive ATOMIC provided the best scalability in the tests. Using this directive, many critical sections, exist being very small and their execution is supported by hardware.

2.2. Parallel solver for the system of equations

The parallel direct sparse solver HSL MA86, also suitable for indefinite matrices, was interfaced to ompFEAP. The source code is available for this solver, the method implemented is described in [4].

3. Numerical results

In order to assess the speedup of computations caused by the OpenMP parallelization of FEAP, calculations were performed for two RVE material models. In both models, the 3D 8-node standard (Lagrangian) finite element was used.

The first model, for a metal-ceramic composite, was obtained...
in [6] from the micro-CT scans, transformed into an FE mesh shown in Fig. 1 using a specialized software. This model involves 0.39 million dofs.

The second model, for a ceramic foam, is shown in Fig. 2. It was developed in [5] to correspond to the micro-structure recorded on CT scans. This model involves 3.8 millions dofs.

The computations were performed on the machine with 2 processors Xeon X5650 2.66GHz (6 cores each) and 24GB DDR3 1333MHz RAM memory. The code was compiled using Intel Compiler ver. 14.0.0, optimization flag 2.

In order to verify the correctness of parallelization, additionally to Intel Inspector 2015 tool, the computed average material parameters were compared to the reference paper results. They were identical regardless of the number of threads used.

Table 1: Total time of parallel parts of ompFEAP and memory usage.

<table>
<thead>
<tr>
<th>Number of threads</th>
<th>Metal-ceramic composite</th>
<th>Ceramic foam</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>325.03</td>
<td>5.47</td>
</tr>
<tr>
<td>12</td>
<td>37.46</td>
<td>6.22</td>
</tr>
</tbody>
</table>

The total time of parallel parts and the memory usage are reported in Table 1. The total time includes the time in the loop over elements (of matrix generation and assembling) and of the solution of the system of equations. Due to the metal-ceramic composite, the time is 8.7 times shorter for 12 threads (parallel execution) than for 1 thread (serial execution) and the memory usage is increased by 14%. Due to the ceramic foam, the time is 8.9 times shorter for 12 threads than for 1 thread and the memory usage increased by 14%.

Additionally, the scalability of particular parts of the parallel code is shown in Fig. 3 for the metal-ceramic composite. According to the loop over elements, which includes the matrix generation and assembling, the speedup is 10.97 while for the solver HSL MA86 the scalability is 8.64, both for 12 threads. The first speedup indicates that no more sophisticated approach to the parallelization of the loop over elements is required.

Figure 1: FE mesh for metal-ceramic composite.

Figure 2: FE mesh for ceramic foam. (Much finer mesh than shown was used in computations.)

Figure 3: Scalability of ompFEAP for metal-ceramic composite.

References

Receptance coupling for turning with a follower rest

Marcin Jasiewicz¹, Bartosz Powałka²

¹,² Faculty of Mechanical Engineering and Mechatronics, West Pomeranian University of Technology Szczecin
19 Piastów Ave., 70-313, Poland
e-mail: marcin.jasiewicz@zut.edu.pl ¹, bartosz.powalka@zut.edu.pl ²

Abstract

The paper presents an issue of dynamic modelling of turned parts with a follower rest for machining stability prediction. Dynamic properties of a spindle, tailstock and the follower rest are assumed constant, to be determined experimentally based on results of impact test. Hence, the variable of system “machine - handle – part – tool” is the machined part and a follower rest setting, which can be modelled analytically. The method of receptance coupling enables a synthesis of experimental (spindle, tailstock, follower rest) and analytical (machined part) models, so the impact testing of the entire system becomes unnecessary. The paper presents methodology to synthesize an analytical model of the machined parts with the spindle, tailstock and follower rest experimental models. In the summary experimental verification of the calculations is presented.

Keywords: turning, follower rest, receptance coupling

1. Introduction

Machining of compliant parts is difficult to carry out due to occurring high – amplitude vibrations. An example of such machining may be slender rods turning, usually implemented using steady or follower rest (Fig. 1) to provide stiffening of the workpiece so that the turning process remains stable.

Figure 1: Turning with a follower rest

In order to assure stable cutting conditions, selection of the cutting depth and spindle speed can be carried out using the so-called stability lobes Ref. [3], which depend on dynamic properties of the system “machine - handle – part – tool”. Considering machining with the follower rest moving along with a tool post, it should be noted that besides the geometry and material properties of the workpiece, the rest location will have a significant influence on these properties.

Experimental determination of dynamic properties of the system for each object mounted on a lathe at different follower rest locations is too laborious, requiring access to the appropriate measuring equipment.

The elements of the system, where the dynamic properties can be considered constant during the machining are spindle, tailstock and follower rest, to be determined experimentally (eg. impact test), or based on the FEM modelling. Furthermore the workpiece can be considered as a circular cross section beam and its dynamic properties could be determined analytically.

Having transfer functions of all system components a synthesis using the method of receptance coupling Ref. [4] can be carried out.

In the References [2,4,5,6] the synthesis of the transfer function is made for the purpose of determining the dynamic properties of milling spindle – tool assembly. In the paper, the method of receptance coupling used for determining the dynamic properties of lathe is presented, where the workpiece is modelled analytically, the spindle, tailstock and follower rest – experimentally.

2. System components

Methodology of determining the transfer function of the “machine - handle – part – tool” system components is presented here.

2.1. Spindle

Dynamic properties of the spindle does not change during machining, so may be determined experimentally (impact test).

Apart from determining the transfer function in x direction it is also necessary to identify rotational degrees of freedom (RDOF) Ref. [5] (Fig. 2) because chuck fixture causes that rotation angles of the spindle and workpiece at the point of attachment are the same.

Figure 2: Lathe spindle with the chuck
RDOF transfer functions can be determined indirectly based on impact testing using the first order finite difference method [1,7].

2.2. Workpiece

Another element of the system is the workpiece. It could be modelled as a free-free beam with circular cross section. The beam is considered to be free, because the boundary conditions are applied to the system during its synthesis and arise from the spindle, tailstock and follower rest properties.

2.3. Follower rest and tailstock

Since rotational motion of the workpiece is independent of rotation of the tailstock and follower rest it is sufficient to determine only translational response functions on the basis of the results of impact tests.

3. Receptance Coupling

Having all required transfer functions of the system components (Fig. 3) synthesis using receptance coupling could be performed.

Figure 3: System components

Imposing the boundary conditions (1) and the equilibrium of forces (2) between the isolated coordinate systems of components and assigning them to a coordinate of a merged system its transfer function could be synthesised.

Boundary conditions:

\[
x_{x1} = x_{x1} = x_1 \\
\phi_{\phi1} = \phi_{\phi1} = \phi_1 \\
x_{x2} = x_{x2} = x_2 \\
x_{x4} = x_{x4} = x_4
\]

(1)

Equilibrium of forces:

\[
F_{x1} + F_{x1} = F_1 \\
M_{x1} + M_{x1} = M_1 \\
F_{x4} + F_{x4} = F_4
\]

(2)

4. Experimental verification

The next step of the research was to conduct experimental verification. The identification of the dynamics of the spindle, follower rest and tailstock were based on the results of impact tests. Using the obtained transfer function and determining the geometry of the workpiece in the analytical model the transfer function of merged system was synthesized. Then, the workpiece was mounted on the lathe and for specific follower rest location a series of impact tests were carried out. Finally the transfer functions obtained experimentally are compared with those previously synthesized.

5. Conclusions

In conclusion the results of the tests are discussed, the applicability of the procedure and determined the direction of further research.

References


A method of estimating the stability coefficient of a considerably degraded cooling tower

Tomasz Kasprzak1, Piotr Konderla2, Ryszard Kutyłowski3, Grzegorz Waśniewski2
1,3,4 Faculty of Civil Engineering, Wrocław University of Technology
wyb. St. Wyspiańskiego 27, 50-370 Wrocław, Poland
e-mail: ryszard.kutyłowski@pwr.edu.pl
2 Technical-Engineering Faculty, Wrocław University of Technology
wyb. St. Wyspiańskiego 27, 50-370 Wrocław, Poland
e-mail: piotr.konderla@pwr.edu.pl

Abstract

The aim of the research is to develop a method of estimating the reduction in the coefficient of stability of a cooling tower caused by wear. As a result of material deterioration Young’s modulus of the concrete and the thickness of the cooling tower shell change, contributing to changes in the stiffness of the shell. This paper presents an algorithm for determining the function of structural stability coefficient sensitivity in the cooling tower shell to changes in shell stiffness, whereby a map of the sensitivity of the cooling tower shell structure is obtained. Having data on the actual degree of degradation of the shell the current stability coefficient of the analyzed cooling tower may be estimated.

Keywords: cooling tower stability coefficient, shell material degradation, methodology for determining safety level of cooling towers with regard to stability

1. Introduction

The design form of cooling towers meets the stability condition usually with a certain margin. Over years of service the materials of the cooling tower structure undergo degradation, whereby their physical parameters, especially those of the concrete, change. As a result of the surface degradation of the cooling tower shell the effective thickness of the latter decreases. Consequently, the cross-sectional stiffness of the cooling tower shell also decreases, lowering the cooling tower structure stability coefficient [1]. Although routine tests supply information about the parameters of the material and the structure, the number of local parameter determinations is small because of the unit costs (considering the large cooling tower surface) of such tests. The aim of the research was to develop a methodology for determining the shell safety level with regard to stability loss on the basis of a small set of in situ test results documenting the degree of cooling tower shell degradation.

A structural safety level with regard to stability is determined solving the initial stability problem for the structure. In particular, the critical load multiplier, called a stability coefficient (\( \lambda \)) [3], is calculated. For a cooling tower the coefficient should meet the condition \( \lambda \geq 5 \) [4]. The critical load level directly depends on the structural stiffness, the latter is the function of two parameters: the Young’s modulus of the concrete and the thickness of the shell wall.

The proposed method of stability analysis was tested on a cooling tower investigated in situ by the authors [2]. The cooling tower had been in service for several decades and its shell showed considerable degradation.

One of the results of the analysis is a function of the sensitivity of the cooling tower to a change in the stability coefficient due to the degradation of the particular cooling tower areas. This function can be used in a similar way as the areas of influence of, e.g., the internal forces. Having such a function and appropriate material test results in discrete points of the cooling tower shell the degree of safety of the cooling tower may be determined with regard to stability loss.

2. Theoretical foundations

It is assumed that in a newly built cooling tower the distribution of shell flexural stiffness is axially symmetric (\( \mathcal{K}(z) \)). For homogenous concrete a change in shell stiffness along the height results from a change in shell thickness. As the material undergoes degradation over time the cooling tower stiffness parameters change. Let \( \mathcal{K}(x) \) represent the current distribution of shell thickness under bending, where \( x = (z, \varphi) \). In order to identify function \( \mathcal{K}(x) \) tests were carried out in \( N \) points of the shell by measuring the thickness of the shell and the Young’s modulus of concrete and on this basis the values of function \( \mathcal{K}(x) \) in points \((x_1, x_2, \ldots, x_N)\) were calculated. The reduction in stiffness in the \( i \)-the point is

\[
\frac{D_i}{K_i} \triangleq \mathcal{K}(z_i) \triangleq \mathcal{K}(x_i). \tag{1}
\]

Similarly to the division into finite elements in FEM, shell surface \( V \) is divided into subareas \( V_i \) where changes in stiffness can be approximated in the form of the series

\[
D(x) = \sum_i D_i N_i(x), \tag{2}
\]

where \( N_i(x) \) performs the role of base (shape) functions while \( D_i \) performs the role of a nodal point of stiffness change approximation. For a proper selection of base functions a function \( D(x) \) is continuous on \( V \).

Let \( \lambda \) be a stability coefficient of the cooling tower if changes in stiffness in \( V_i \), in the form

\[
K_i(x) = K_0(z) - \Delta K \cdot N_i(x), \tag{3}
\]

where \( \Delta K \) is a fixed small quantity relative to \( \min\{K_0(z)\} \), are superimposed on the ideal geometry of the shell.

The difference \( \lambda - \lambda_0 \) is a measure of stability coefficient sensitivity to shell stiffness function in area \( V_i \), with distribution \( \Delta K \cdot N_i(x) \). The notion of sensitivity \( \eta(x) \), understood as the in-
tensity of stability coefficient change as a result of a change in shell stiffness in a unit surface area by a unit value, is introduced. This function can be presented in a base \( \{N(x)\} \)

\[
\eta(x) = \sum_j \eta_j \cdot N_j(x) .
\]  

(4)

Then the nodal values in equation (4) are equal to

\[
\eta_j = \frac{\lambda_0 - \lambda_j}{\Delta K} \left[ \int V N_j(x) dV \right]^{-1}.
\]  

(5)

For a set field of the cooling tower shell stiffness change, the current stability coefficient at arbitrary time instant \( T \) can be calculated from the equation

\[
\lambda_T = \lambda_0 - \int D(x) \cdot \eta(x) dV .
\]  

(6)

3. Calculation example

A cooling tower was analyzed, its static diagram is shown in Fig. 1a. The shell parameters were tested on 11 latitudinal levels. A calculation test was carried out for cooling tower concretes with Young’s modulus of 31.4 GPa. Table 1 shows how much the Young’s modulus and shell thickness values decreased on four successive levels of the cooling tower (counting down) as a result of material degradation. On the other levels the Young’s modulus and shell thickness values remained unchanged. Assuming a constant change in stiffness relative to the design stiffness, discrete distribution \( D(z) \) of shell change was obtained as shown in Fig. 1c. The base functions were defined bilinear along the generating line and constant along the circumference, whereby a continuous distribution of function \( D(x) \) within cooling tower shell area \( V \) is obtained.

Determining the stability coefficient \( \lambda_0 \) for the cooling tower design parameters and cooling tower stability coefficients \( \lambda_1, \lambda_2, ..., \lambda_{11} \) the sensitivity function shown in Fig. 1b was obtained for the next stiffness fluctuations according to stiffness distribution (2). In order to determine the influence of the measured stiffness parameters of the analyzed cooling tower on the lowering of the stability coefficient the expression (6) is used. For the considered case, after transformations it reads

\[
\Delta \lambda = \int D(x) \cdot \eta(x) dV = \sum_j \int V N_j(x) \cdot N_j(x) dV = 0.352 .
\]  

(7)

It appears that due to shell material degradation the stability coefficient from the initial value \( \lambda_0 = 6.260 \) decreased to \( \lambda_T = 5.908 \). The obtained result was verified by determining stability coefficient \( \lambda_T \) on the basis of the measured changes. In this case, \( \lambda_T = 5.897 \pm \lambda_r \) was obtained, validating indirectly the proposed methodology.

### Table 1: Values by which stiffness parameters decreased

<table>
<thead>
<tr>
<th>level</th>
<th>( \Delta E ) [MPa]</th>
<th>( \Delta h ) [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>300</td>
<td>0.025</td>
</tr>
<tr>
<td>2</td>
<td>2000</td>
<td>0.020</td>
</tr>
<tr>
<td>3</td>
<td>1500</td>
<td>0.015</td>
</tr>
<tr>
<td>4</td>
<td>1000</td>
<td>0.010</td>
</tr>
</tbody>
</table>

4. Conclusion

Having proper sets of laboratory-determined cooling tower shell Young’s modulus values and in-situ cooling tower shell thickness test results and using the proposed algorithm the actual degree of safety of the cooling tower shell with regard to stability loss may be determined. The proposed methodology opens up possibilities for the synthetic analysis of the stability problem, in particular for: (a) determining the shell areas the degradation of which significantly contributes to the lowering of the stability coefficient and (b) estimating the stability coefficient as a random variable (treating the test results as random variables). Considering the geometrical similarity between a large number of cooling towers built in the 1970s and 1980s, the sensitivity functions for the structures should be qualitatively very similar.

### References


Wave propagation for diagnostics of connections in steel structures

Rafal Kędra¹, Magdalena Rucka²*

¹,² Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: rafkedra@pg.gda.pl; mrucka@pg.gda.pl

Abstract

The paper presents results of numerical and experimental investigations of elastic wave propagation in a bolted single-lap joint. The research was carried out for both the excitation in and out of the connection plane (symmetric and antisymmetric Lamb wave modes). In experimental studies the PZT transducers were used to excite and register wave voltage signals. The controlled value of the torque during measurements was ensured by the use of high precision torque wrenches. Numerical model of the joint was developed by the finite element method commercial software Abaqus. The analysis took into account varying contact conditions between the elements of the connection depending on the value of the bolt torque. Both experimental and numerical results showed the influence of the value of pretension force to amplitude and phase shifts of registered signals.

Keywords: bolted lap joint, wave propagation, non-destructive diagnostics, experimental investigations, finite element method

1. Introduction

Bolted joints due to their high load carrying capacity, durability and easy installation are very popular, so often used in many industrial sectors, including numerous applications in construction industry. The prestressed bolt connections recently gained popularity. A disadvantage of this type of connection is the possibility of load capacity decrement with time due to self-loosening and rheological effects. For this reason in the case of joints crucial for the overall structural stability there is a need to develop methods of non-destructive diagnostics and damage detection in early stages of their development.

A possible strategy of diagnostics of bolted connections, intensively developed in recent years, is associated with elastic waves phenomenon [1,2,3,4,5,6,7,8,9,10]. Qualitative and quantitative indicators based on the variability of recorded signals are used in the state assessment. In many cases, the research on those diagnostic methods was connected with the development of analytical models describing wave propagation. The main difference lies in the solution of the problem of interaction of the contact conditions between the connected elements. Different modelling approaches were used to describe propagation at the interface surface. A detailed analysis of the effect of pressure on the contact condition of composite panels in micro-scale was carried out by Yang and Chang [9,10]. The authors have shown a relationship between a nominal and true contact area for sinusoidal wavy surface using Hertz contact theory. Rhee et al. [6] created a relatively simple model of a multi-bolt lap joint.

They analyzed two models of bolts (solid and rigid) and took into account contact between bolts and plates by a constrained lateral surface of bolt stud with the edges of the holes in the plates. In addition, they created an accurate model of piezoelectric transducers. Finally, the authors were able to reproduce accurately the shape of the experimentally recorded signals. They concluded low impact of prestressing force on the variability of recorded waveform. Huda et al. [4] used a multi-step approach. They assumed a circular shape of a contact area and to determine its radius, by means of static analysis. Afterwards, in the second step they assumed constant contact conditions between the bolt and plates. Furthermore, they introduced the stress and strain results obtained in static analysis as initial conditions in wave propagation analysis. As a result, the authors obtained very good agreement of dynamic parameters of the developed model and experimental data for the cases of one- and multi-bolt joints.

The models described above are relatively simple and they are intended to reproduce results of a single experiment. Therefore, they do not include the full complexity of the phenomenon of wave propagation. There is still a need to develop a more accurate model, able to cover all physical aspects related to wave propagation. The aim of the study is to develop a model of wave propagation using finite element software Abaqus. Numerical studies were performed in parallel with experimental tests. The obtained results were compared and evaluated in terms of a possible application in diagnostics of bolted lap joints.

2. Studies of single lap bolted joints

The research was conducted for the models of single lap bolted joints. They were made of two steel plates, which were assembled together by one bolt of a nominal diameter of 12 mm and a length of 49.5 mm, nut and two stainless steel washers. The tests were performed for two kinds of plates, with dimensions 6.4 mm × 40.4 mm × 440.5 mm and 7.8 mm × 40 mm × 438.5 mm. The geometry of the connection is shown in Fig. 1.

![Figure 1: Scheme of analyzed bolted connection](image-url)
2.1. Experimental tests

During experimental analysis the model of a bolted joint was located in a steel frame to ensure a constant position of steel plates during the bolt tightening process. Excitation and acquisition of elastic voltage signals were carried out by the device PAQ-16000D and piezoelectric transducers Noliac NAC2024. The sensors and actuator were attached to the model at selected points with the use of wax. The excitation had forms of a sine function modulated by the Hanning window. The sampling frequency during the measurements was set as 2 MHz.

2.2. Numerical analysis

Numerical analysis of wave propagation at joints was performed using the finite element method (FEM) program Abaqs. Additionally, to reflect accurately all elements of models, the geometry of joints was developed in AutoCAD software. All parts of the joint and sensors were discretized using solid brick elements (six-node and eight-node) with reduced integration and maximum edge size equal to 1 mm (Fig. 2). The fixed-fixed boundary conditions of bolted joints were assumed. Contact between elements of the connection was modelled introducing a thin layer of brick elements. Numerical calculations were carried out in several stages. Initially, the geometrical and material nonlinear static analysis was made to determine the contact conditions between the plates, nuts and bolt heads. In the second step wave propagation analysis was simulated. The wave was induced applying an electric field to finite elements, modelling a piezoelectric transducer. The time-variation of excitation was identical to the value from experimental studies. The time step of numerical explicit integration was assumed as 10 ns to ensure stability of the central difference scheme.

Figure 2: Visualization of the finite element mesh

3. Results and conclusions

The results of analysis were presented as voltage and acceleration signals in both time and frequency domains. Additionally, in the case of numerical calculations wave propagation was shown in the form of two-dimensional acceleration maps.

The experimental results indicated an influence of a bolt torque value on the registered voltage signals. With the increase of pretension value, the phase shifts and growth of the amplitude in initial part of signals is observed (Fig. 3). This effect was also confirmed in numerical analyses. However, numerical and experimentally registered signals were comparable only in the initial part of signals. The results showed the possibility of using quantitative characteristics of signals for the purposes of diagnostics of bolted lap joints.

Figure 3: Example of voltage signals registered experimentally for different values of bolt torque

References


Dynamics of underactuated mechanical systems in control task

Sebastian Korczak
Faculty of Automotive and Construction Machinery Engineering, Warsaw University of Technology
Narbutta 84, 02-524 Warszawa, Poland
e-mail: sebastian.korczak@simr.pw.edu.pl

Abstract

The document presents a formulation of coupling input problem in dynamics of underactuated mechanical systems. Novel method of all system accelerations control with pseudoinverse operation and computed torque algorithm was investigated. Numerical simulation shows that this full control method gives better control effects than typical selective control techniques. Analysis dynamic equation of errors while tracking simple trajectory results in highly complex and coupled ODEs.

Keywords: control, underactuated system, coupling force, tracking

1. Introduction

The dynamic system described by a set of second order ordinary differential equations in form

\[
\ddot{q}(q, \dot{q}, t) = f_2(q, \dot{q}, t) + f_3(q, \dot{q}, t)u(t)
\]

where generalized coordinates

\[
q(t) = [q_1(t), q_2(t), \ldots, q_n(t)]^T
\]

and input forces

\[
u(t) = [u_1(t), u_2(t), \ldots, u_m(t)]^T
\]

called underactuated if unbounded control inputs \(u(t)\) cannot produce accelerations \(\ddot{q}\) in arbitrary direction. This property can be verified by a condition

\[
\text{rank}[f_2] < \text{dim}[q]
\]

Usually there are trivially underactuated systems, where number of inputs is less than the number of degrees of freedom.

There are many typical systems with underactuated property, e.g. cart-pole, acrobat, pendubot, airplanes and multicopters, hovercrafts, surface vessels and underwater vehicles. Current review of underactuated systems and their control is presented in [4].

2. Description of the system

In this section one of the simplest underactuated model is described [2]. It can be used for basic representation of hovercraft, rocket or sliding vehicle.

Figure 1: Object in the global coordinate system

2.1. Physical model

Consider a planar rigid body moving on a plane (Fig. 1). The object has mass \(m\) and inertia \(I_c\) in a centre of mass (point \(C\)). Coordinates \(x(t)\) and \(y(t)\) describes its position, \(\varphi(t)\) denotes angle between the object symmetry line and \(X\) axis of the global coordinate system \(O_{xy}\). The vector of force \(F\) acts on the object in a point away from the point \(C\) by distance \(a\). Constant drag coefficients, \(c\) for linear motion and \(c_\varphi\) for rotation, are used. The equations of motion for the system are as follows

\[
m\ddot{x}(t) + c\dot{x}(t) = F(t) \cos(\varphi(t) + \beta(t))
\]

\[
m\ddot{y}(t) + c\dot{y}(t) = F(t) \sin(\varphi(t) + \beta(t))
\]

\[
I_c\ddot{\varphi}(t) + c_\varphi\dot{\varphi}(t) = F(t) a \sin \beta(t)
\]

2.2. Controllability

After rewriting Eqns (3-5) into matrix form of the first order ODEs, small-time local controllability of the system can be proofed (accessible algebra rank condition is satisfied) [2].

2.3. Trajectory tracking

Let us consider two trajectory tracking tasks: circular and eight curve motion with tangential orientation described by desire functions

\[
q_{da}(t) = \begin{bmatrix} R \cos \theta t \\ R \sin \theta t \\ \theta t + \theta_0 \end{bmatrix}
\]

\[
q_{da}(t) = \begin{bmatrix} 0.5R \sin 2\theta t \\ R \sin \theta t \\ \text{Arg}(\cos 2\theta t + \sqrt{1 - \cos \theta}) \end{bmatrix}
\]

where \(R\) is trajectories’ size parameter and \(\theta\) is the angular velocity. Trajectory tracking, path following and stabilization problems of underactuated systems are usually solved with backstepping technique [1], sliding mode [6], velocity filed or flatness-based methods.
3. Input coupling problem

One of the problems of underactuated systems problem is input coupling. The system described by Eqn (1) has input coupling if there does not exist a combination of basic row operations that converts it into a form [5]
\[ \tilde{q}(\tilde{q}, \dot{q}, \ddot{q}, t) = f_2(\tilde{q}, \dot{q}, t) + f_1(\tilde{q}, \dot{q}, t)u(t) \] (8)
where
\[ f_2 = \begin{bmatrix} f_{21} \end{bmatrix}, \quad f_{21} \equiv 0 \quad \text{and} \quad \text{rank}[f_{22}] = \text{dim}(q) \]

The most popular method of control of underactuated systems with input coupling effect is based on change of variables that converts problem into noncoupled. Then some accelerations are separately controlled by inputs and some stayed without a control (selective control method).

The problem of all accelerations control (full control method) not solved properly yet can be solved using Moore-Penrose pseudoinverse operation combined with computed torque technique and PD feedback [3]. Inputs are then proposed as
\[ u(t) = f_2^T f_4 \] (9)
where pseudoinverse of a real numbered rectangle matrix can be obtained using formulation
\[ f_2^T = \lim_{\delta \to 0} (f_2^T f_2 - \delta I)^{-1} f_2^T \] (10)
and desired accelerations
\[ \ddot{f}_d = \ddot{q}_d - f_1 + K_0(q_d - \dot{q}) + K_p(q_d - q) \] (11)

4. Behaviour of tracking errors

Tracking errors are defined as differences between corresponding desired trajectory functions and real system coordinates as follows
\[ e(t) = q_d(t) - q(t) \] (12)
For the system presented at this contribution, an error matrix has a form
\[ e = [e_x(t), e_y(t), e_\phi(t)]^T = [x_d - x, y_d - y, \phi_d - \phi]^T \] (13)
Substitution of control method proposed by Eqns (9-11) into system equation of motion (1) cause errors dynamic equation
\[ \ddot{e} + f_2 f_2^T K_\text{pe} + f_2 f_2^T K_\text{pe} e = (f_2 f_2^T - I) (f_1 - \ddot{q}_d) \] (14)
with \( f_2, f_2^T \) and \( f_1 \) being the functions of \( q \).

4.1. Circular trajectory

Figure 2 presents a result of numerical simulation of circular trajectory tracking. Pseudoinverse algorithms cause unreal convergence to zero errors in place of all accelerations control.

4.2. Eight shape trajectory

Figure 3 presents exemplary result of numerical simulation of eight-shaped trajectory tracking with same good results.

5. Conclusions

This contribution presents a formulation of coupling input problem in dynamics of underactuated systems. Novel method of all system accelerations control with pseudoinverse operation and computed torque algorithm was proposed. Stability analysis of underactuated systems with input coupling controlled by pseudoinverse algorithm is still and open problem due to Eqn (14) form. Numerical simulation shows that a full control method gets better control effects than typical selective control techniques.

References


A numerical analysis of the ultimate strength of longitudinally unstiffened girders subjected to patch loading

Sasa Kovacevic¹, Nenad Markovic²

¹Energoprojekt Industrija, Bulevar Mihaila Pupina 12, 11070 Belgrade, Serbia
e-mail: s.kovacevic@ep-industry.com
²Faculty of Civil Engineering, University of Belgrade, Bulevar kralja Aleksandra 73, 11000 Belgrade, Serbia
e-mail: nenad@grf.bg.ac.rs

Abstract

In the paper results of numerical research of the ultimate strength of longitudinally unstiffened steel girders due to patch loading are presented. Numerical tests are compared to results of experimental tests on welded I-shaped girders using different patch load length. Patch loading strength was performed by nonlinear analysis including geometrical and material nonlinearities. The real stress-strain diagram was implemented in FE model, these results were compared with tests using simplified stress-strain curves. Besides, the FE model contains the real initial geometric imperfections based on the laboratory tests, these shapes were compared with sine shapes. Numerical simulations were carried out using a commercial multi-purpose FE analysis software package Abaqus.

Keywords: patch loading, numerical simulation, ultimate strength, plate girder, experimental tests, geometrical imperfections

1. Introduction

Patch loading phenomenon is a special load case where a concentrated or partially distributed load is applied on one flange over a distance called patch load length. The behaviour of steel plate girders under patch load represents complex stability and elasto-plastic problems [3]. This load acts in the plane of a web without vertical stiffener bellow the load. This situation appears in the case of the moving load and it is present in phase of launching bridge girders or in crane girders. Following this and general trend for avoiding vertical stiffeners except at the supports, this problem is of great importance.

This work presents a comparison between experimental investigation on welded I-shaped girders and numerical research applying different patch load length. The results from numerical tests, which include accuracy and convergence for patch load resistance are compared with experimental results. The real initial geometric imperfections (measured before the experiment) and sine shapes (both in the transverse and the longitudinal direction) with maximum allowable amplitude of 5 mm according to EC 3-Part 1-5 were considered. Furthermore, different types of stress-strain diagrams were studied. Beside real stress-strain diagram (Fig. 3) the simplified curves with or without strain hardening or necking were examined.

2. Laboratory test

The experimental research includes three tests using different patch load length, c=0, c=25, c=50 mm. These girders are labelled as A15, A12 and A1 in Ref. [2]. The load was applied at midspan, centrically over the web, on the upper flange. For test set-up see Fig. 1. For experimental results and geometrical properties of the girders see Tab. 1.

3. Numerical test

For simulation of numerical model commercial multi-purpose FE analysis software Abaqus was used. Patch loading resistance of the steel plate girder was performed by incremental nonlinear analysis using geometrical and material nonlinearity. Nonlinear static equilibrium states during the unstable phase of the response can be determined by modified Riks method which has been implemented in Abaqus [1]. This method is incremental-iterative procedure and it is suitable for predicting unstable, geometrically nonlinear collapse of a structure including nonlinear materials and boundary conditions.

The girders were modelled in full size using four node quadrilateral shell elements with reduced integration S4R, Fig. 2. Finite element meshes were developed using element size of 20 mm to 6 mm. To implement the real loading conditions the
Table 1: Experimental results and geometrical properties

<table>
<thead>
<tr>
<th>Girder</th>
<th>b_f [mm]</th>
<th>t_f [mm]</th>
<th>l_m [mm]</th>
<th>h_w [mm]</th>
<th>b [mm]</th>
<th>c [mm]</th>
<th>F_{exp} [kN]</th>
</tr>
</thead>
<tbody>
<tr>
<td>A15</td>
<td>120</td>
<td>8</td>
<td>4</td>
<td>500</td>
<td>500</td>
<td>0</td>
<td>143.30</td>
</tr>
<tr>
<td>A12</td>
<td>120</td>
<td>8</td>
<td>4</td>
<td>500</td>
<td>500</td>
<td>25</td>
<td>154.60</td>
</tr>
<tr>
<td>A1</td>
<td>120</td>
<td>8</td>
<td>4</td>
<td>500</td>
<td>500</td>
<td>50</td>
<td>165.00</td>
</tr>
</tbody>
</table>

Load was applied at the midspan on the upper flange. The loaded area has width b_f and length c. These nodes (across flange width) were also restrained in the x direction (planes perpendicular to the girder axis) and in the direction of the girder axis. Next, at the upper flange one node at each end were restrained in the x direction.

The stress-strain diagrams were implemented in FE models and these results were compared. Material was modelled as an isotropic material with von Mises yield criterion. Young's modulus and Poisson's ratio were accepted 205 GPa and 0.30 respectively. The yield stress and hardening characteristics were taken from tensile test of girder A12 with values for f_yw=f_yf=321 MPa, Fig. 3.

4. Results and conclusions

Possibilities for numerical modelling of I-shaped steel girders subjected to patch loading were demonstrated in the work. Patch loading resistance was defined using finite element analysis with shell elements S4R and varying element size. Numerical simulations were calibrated with three experimental tests using different patch load length. Relative differences between experimental and numerical results for ultimate load for all three girders using different stress-strain diagram and different initial imperfections are displayed. The following remarks should be pointed out from the presented research:

- experimental value of ultimate load is slightly greater than numerical value and maximum relative error of 19.95%, 13.76% and 7.83% is noticed in case of real initial imperfections and real stress-strain diagram for A15, A12 and A1 respectively, Fig. 4;
- real initial imperfection can be taken into account using sine shape with amplitude of 5 mm. In this case the maximum relative error for A15, A12 and A1 is 8.42%, 4.32% and 3.06% respectively, Fig. 5;
- the usage of element size of 15 mm the relative error of 4.97%, 6.22% and 1.12% is presented for A15, A12 and A1 respectively for real initial imperfections or 4.77%, 1.11% and 1.06% for sine shapes.

References

Modelling and numerical analysis of GFRP composite panels of bridge deck

Maciej Kulpa¹, Tomasz Siwowski²

¹,² Faculty of Civil and Environmental Engineering and Architecture, Rzeszow University of Technology
Poznańska 2, 35-084 Rzeszow, Poland

e-mail: kulpa@prz.edu.pl, siwowski@prz.edu.pl

Abstract

The paper presents the key assumptions and the result of numerical analysis of the composite panels behaviour when under a static load. The specificity of the structural materials - composite laminates – needs a specific approach to modelling the behavior of the panel on a macroscopic scale. The final parameters of the laminates are determined on the basis of the construction of the laminates on the micro-level (micromechanics models), their elastic parameters and the assumptions of the classical lamination theory. Since the panels were analysed as the elements of a vehicular bridge deck, the LM1 load model was used both in the serviceability limit state check (limit of the maximum deflection) and in the ultimate limit state check (carrying capacity). The failure of the laminates was determined on the basis of two special criteria dedicated to composite materials: maximum stress and Tsai-Wu criterions.

Keywords: composites, road decks, laminates, fiberglass, classical lamination theory, failure criterion, Tsai-Wu, maximum stress

1. Introduction

The paper contains a description of the procedure applied to analyse the behavior of the composite panels. Eventually, the panels are planned to form a slab deck in a vehicular girder bridge. The span of 2.4 m in a simply supported system was adopted as a typical span length. The panels will be manufactured by the Vacuum Assisted Resin Transfer Molding (VARTM) method.

Beside the dead weight of the panels and the road surface elements, a load model was the LM1 Model according to PN-EN 1991-2 [1]. The design criterion for a panel was the maximum deflection under the characteristic load of L / 300 (in this case 8 mm).

After the literature and already existing solutions had been reviewed, three different arrangements of webs were selected for the numerical analysis (Fig. 1).

Figure 1: The cross-section of three arrangements of webs

The panel structure comprised outer sheets, inner webs (stiffeners) and a foam core. Materials engineering was also included in the project, which came down to the choice of reinforcing fibres number and orientation in the certain elements of the deck panels.

The final stage was the analysis of the strength of the critical panel parts under a design load. The analysis was carried out with two failure criteria dedicated to the analysis of composite materials: maximum stress criterion and Tsai-Wu criterion.

2. Lamina level

The structure of panels consists of two types of fabric with glass fibres which are surrounded with epoxy resin. The fabrics, also known as NCF (non-crimp fabric), were distinguished by their stretched fibres inside the individual layers. The fibres in the laminate were placed in mutually perpendicular directions:
- longitudinally and perpendicularly to the panel’s axis (0° / 90°);
- at the angle of ±45° to the panel’s axis (±45°).

Figure 2: The schematic arrangement of the laminae divided with bi-directional fabrics

The assumption was made that the laminae of thickness t, reinforced with the bi-directional fabric, is divided into two parts, which is schematically shown in Fig.2.

Generally, elastic properties of composite materials are defined by elastic modulus (E), the Poisson’s ratios (ν) and shear modulus (G) in three directions. The experimental determination of many properties is a prolonged and costly method. The homogenisation techniques were applied to estimate the elastic properties of the resulting composite on the basis of the elastic properties of the components: the matrix and the fibres. Wherein, the relative volume of the fibre and the matrix was assumed in advance at the level of 53%, which was previously the mean value obtained for laminates with a similar structure and for tests of laminates in the initial phase.

It was assumed that both the fibres and the matrix are isotropic materials on the micro-level. Analytical micromechanical models were used for modelling of the microstructure. The assumption was made that laminae have the
periodic microstructure. At this stage it is assumed that the fibres are uniformly distributed. Based on that, the Fourier series could be used to define the stiffness tensor $C^*$ with six independent components [3]. The engineering constants for resin, fibres and bi-directional laminae are shown in Tab. 1. Inevitable disturbances occurring in a real structure of the composite were taken into account by the rotation of tensor $C^*$ on the axis of fibres and the calculation of a mean value which results in new tensor $B$. Finally, engineering constants were obtained on the basis of the components of tensor $B$ and expressed in terms of components $C^*$.

Knowing the lay-up of laminae, the areal weight of fabric $g_r$ and the density of fibres $\rho_f$, the thicknesses of the laminae $t$ could be determined on the basis of Eqn (1) as in Ref. [2].

$$ t = \frac{g_r}{\rho_f} v_f $$

(1)

The analysis of the strength of the composite was performed at the level of each divisional unidirectional lamina according to the classical lamination theory for thin plates. This implies the omission of the effect of transverse shear strains $\sigma_{xz}$ and $\sigma_{yz}$ and normal strain $\sigma_z$. This approach requires the determination of the composite's strength in reference to its material axis. The lamina’s strength was determined for typical samples of 2 mm thick laminate reinforced with unidirectional glass fibres.

Table 1: The resultant engineering constants for components and the individual unidirectional laminae (on materials’ axis)

<table>
<thead>
<tr>
<th></th>
<th>Elastic modulus</th>
<th>Shear modulus</th>
<th>Poisson’s ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$E_1$ [GPa]</td>
<td>$E_2$ [GPa]</td>
<td>$G_{12}$ [GPa]</td>
</tr>
<tr>
<td>Fibers</td>
<td>73.0</td>
<td>73.0</td>
<td>29.2</td>
</tr>
<tr>
<td>Matrix</td>
<td>2.9</td>
<td>2.9</td>
<td>1.1</td>
</tr>
<tr>
<td>Laminae</td>
<td>40.0</td>
<td>8.3</td>
<td>3.3</td>
</tr>
</tbody>
</table>

3. Laminate level

The linear laminated plate theory, also commonly known as the classical lamination theory, does not take into account the problem of the interplanar shear between the laminae. By operating on stiffness matrices for the laminae, the stiffness matrices for the entire laminate were determined:

- **A** - the in-plane stiffness matrix;
- **B** - the coupling stiffness matrix;
- **D** - the flexural stiffness matrix.

All three matrices are combined into a single stiffness matrix of the laminate. The inversion of the stiffness matrix for the entire laminate enables the calculation of the final compliance matrix for the laminate. As before, the compliance matrix may be divided into components: the in-plane, the coupling and the flexural compliance matrix. Engineering constants obtained in this manner related to two behaviors: in-plane and flexural.

For the strength analysis a complete set of engineering constants (in-plane and flexural) was applied. However, for the description of the substitute material in numerical models and the static analysis only in-plane engineering constants were applied, determined on the basis of normalized in-plane compliance matrix of the laminate. The examples of engineering constants for laminates in panel 1 are shown in Tab. 2. It is important that subscripts "x" and "y" refer to the elements axis, in contrast to "1" and "2", which refer to the materials (fibres) axis.

4. Element level

For the purposes of the static analysis, models in the macrosopic scale were made. The modelling task was limited to one simply supported span. The dimension in the transverse direction was adopted at 1.2 m due to the dimensions of the load model LM1 (one axis of the vehicle is located on the panel). Composite laminates were modelled in Sofistik software code with the use of the 4-node plate elements according to Mindlin’s plate theory and with an extension of a non-conforming formulation. The total rotation is the sum of the shear deformation and the bending rotation (First-Order Shear Deformation). This means that the results obtained directly under the place of the load application (elements loaded transversely) may require more careful analysis [4]. However, this problem does not affect the global behavior of the structure and, due to the thickness of the laminate, can be neglected.

Additionally, the isotropic material and the 8-node solid elements were used to model the foam cores, assuming that these were infinitely firmly bound to the outer sheets. The equal arrangement of elements has been used. As an example, the numerical model of panel 1 is presented in Fig. 3 (9792 plate elements and 25 344 solid elements).

Figure 3: The finite elements model of panel 1: the general view (left) and the deformation of composite elements under the point load of the load model’s wheel in a half span (right).

The final confirmation of the correctness of the adopted panel construction was to verify the capacity of laminates under internal forces estimated in the static analysis.

References

GPR simulation for diagnostics of reinforced concrete structures

Jacek Lachowiez¹, Magdalena Rucka²
¹,²Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mails: jaclacho@pg.gda.pl ¹, mrucka@pg.gda.pl ²

Abstract

The most popular technique for modelling of an electromagnetic field, the finite difference time domain (FDTD) method, has recently become a popular technique as an interpretation tool for ground penetrating radar (GPR) measurements. The aim of this study is to detect the size and the position of damage in a reinforced concrete beam using GPR maps. Numerical calculations were carried out using the finite difference time domain method. Four different damage scenarios with different crack width and shape were investigated. The influence of the frequency of applied electromagnetic (EM) waves on the possibility of damage detection was shown.

Keywords: non-destructive diagnostics, electromagnetic waves, ground penetrating radar, reinforced concrete structures

1. Introduction

The damage assessment of existing structures is of great importance to improve their reliability and safety. The actual state of a structure can be assessed in the process of diagnostics. Recently, various non-destructive testing techniques have been used for evaluation of engineering structures, for example acoustic emission method [8], ultrasonic method [11] or acoustic and vibroacoustic methods [9]. Particularly useful in diagnostics of concrete or masonry structures is the ground penetrating radar (GPR) method [1–4, 10, 12], which uses the phenomenon of electromagnetic (EM) wave propagation.

A primary problem in the GPR technique is the interpretation of measurement data because of a variety of factors that can affect GPR signals. In order to make an appropriate analysis of GPR data, a considerable amount of expertise in interpreting of experimental results is required. Moreover, different defects can provide similar types of patterns in GPR maps. Numerical modelling of electromagnetic wave propagation can provide a significant support in understanding the origin of reflections appearing in GPR experimental data. Moreover, numerical analyses allow performing a preliminary analysis, with different damage scenarios prior to the experimental studies.

2. Problem formulation

The aim of the study is to detect the size and the position of damage in a reinforced concrete beam using GPR maps. The beam was 5m long, a rectangular cross-sectional dimensions 30 cm and 40 cm. The scheme of the analyzed beam is shown in Fig. 1. Damage was considered as an open crack of rectangle, triangle and trapezoid shape. Four different damage scenarios with different values of crack width were investigated. Numerical calculations were carried out using a program gprMax 2D [7]. The Maxwell’s equations were solved using the finite difference time domain (FDTD) method with appropriate boundary and initial conditions in two-dimensional space. Calculations were performed on the middle part of the beam span over a distance of 2 m. The influence of different frequencies of electromagnetic waves on resulting radargrams was examined.

Figure 1: Model of the reinforced concrete beam with different damage scenarios

3. Numerical analysis

The Maxwell’s equations form a theoretical basis for electromagnetic wave propagation. They express the mathematical relationship between magnetic and electric fields, charges and currents. In a lossy and anisotropic medium, the Maxwell’s equations take the form [7]:

\[ \nabla \times \mathbf{E} = \frac{\partial \mathbf{B}}{\partial t}, \]

\[ \nabla \times \mathbf{H} = \frac{\partial \mathbf{D}}{\partial t} + \mathbf{J}_c + \mathbf{J}_s, \]

\[ \nabla \cdot \mathbf{B} = 0, \]

\[ \nabla \cdot \mathbf{D} = q_v, \]

where: \( \mathbf{H} \) and \( \mathbf{E} \) – magnetic and electric field strength vector, \( \mathbf{B} \) – magnetic flux density vector, \( \mathbf{D} \) – electric displacement vector, \( \mathbf{J}_c \) – conduction current density, \( \mathbf{J}_s \) – impressed current density, \( q_v \) – volume electric charge density.
FDTD method [5, 6] developed by Yee [13]. The approach of this method is to discretize both the time and space continua. The numerical model is composed of Yee cells. To set a material, constitutive parameters (permeability $\mu$, permittivity $\varepsilon$ and conductivity $\sigma$) are assigned at nodes of Yee cells. At the border of computational space, absorbing boundary conditions (ABC) are applied which are designed to simulate propagation of the EM wave indefinitely.

The results of calculations for the intact beam and the beam with damage (in the form of 5 mm rectangular crack) are shown in Figs. 2 and 3, respectively. In both GPR maps, called radargrams, several parabolas are visible. These parabolas correspond to reflection form reinforcement bars. Additional parabola present in the lower part of the map shown in Figure 3 corresponds to reflection from damage.

![Figure 2: Calculated radargram for the beam without damage for GPR antenna with 2 GHz frequency](image1)

![Figure 3: Calculated radargram for the beam with damage for GPR antenna with 2 GHz frequency](image2)

4. Conclusion

The results of numerical simulations of EM wave propagation in concrete structures with different damage scenarios are very desirable in the interpretation of GPR maps. Simulations carried out in the paper showed a possibility to detect different types of open cracks in the reinforced concrete beam. The influence of the frequency of the applied EM waves on the possibility of damage detection is shown.

References


Abstract

The numerical analysis focused on reduction of vibrations of a temporary steel scaffolding grandstand has been conducted in this paper. These types of structures are regularly subjected to dynamic loads which, in conjunction with light and quite slender structural members, may induce dangerous vibrations. To increase their safety, temporary steel grandstands are usually strengthened with the diagonal stiffeners of a tubular cross section. Another approach, using a diagonal element consisting of two L-shaped steel members bonded with polymer mass, has been considered in the present paper. The first stage of the study has been devoted to modal analysis. Dynamic parameters, such as modes of free vibrations and the corresponding natural frequencies for both models, were estimated and compared. In order to verify the effectiveness of the polymer element, the dynamic analysis has been conducted. Behaviour of temporary steel grandstand under crowd load has been obtained and peak values of accelerations and displacements for both load cases have been determined and compared. The results of the study show that the responses of the temporary steel grandstand equipped with the polymer element as well as with the typical stiffener are substantially different. The application of the polymer element leads to substantial reduction in the level of measured accelerations due to increased structural damping.

Keywords: dynamic load, steel grandstand, polymer element, crowd load

1. Introduction

A large number of sporting, music and other entertainment events are organized nowadays around the world. Temporary steel grandstands erected with scaffolding system are often used during such events. These structures are regularly subjected to dynamic loads which, in conjunction with light and quite slender structural members, may induce dangerous vibrations. Human-induced vibrations may cause serious problems, including damages or collapses of structures and panic among people. Previous numerical studied confirmed that mass of spectators significantly decreases values of natural frequencies making the structure more vulnerable to structural damage.

One of the most common methods used to reduce the grandstand vibrations is the application of a bracing system. It consists of additional diagonal elements installed on the structure to increase its stiffness. In this paper, an alternative method of reduction of structural vibrations is considered. An additional element that consists of two L-shape steel members (50x50x5 mm) bonded with polymer mass of thickness 5 mm is analyzed. The element was installed as a diagonal one at the back part of the structure. The polymer mass is a specially designed flexible two-component grout, which is based on polyurethane resin. It has been proven that this kind of material has high damping properties.

The aim of the paper is to analyse numerically the response of the temporary steel scaffolding grandstand subjected to dynamic loads due to jumping. Changes in the level of human-vibration perception and comfort are considered.

2. Numerical analyses

Two different numerical models of a part of a typical temporary steel grandstand have been generated using commercial programme MSC Marc. The first of them describes the structure with typical stiffener member of tubular cross-section (Model 1), while the second one concerns structure equipped with polymer element (Model 2). The empty grandstand and the structure that takes into account mass of spectators was considered in the study. That Twelve people were assumed occupy the structure.

A scaffolding system consisting of slender tubular structural member was modelled by standard eight-node (six degrees of freedom for each node) solid elements have been used. The base of the models was considered to be fixed only in translational directions. The numerical models for empty temporary steel grandstand with typical stiffener member of tubular cross-section as well as with polymer element are shown in Figure 1a and Figure 1b, respectively.

The behaviour of polymer was simulated with the use of the Mooney-Rivlin material model that is the most frequently adopted method for modelling complex mechanical behaviour of elastomers and rubber-like solids. The following material constants for the five-parameter Mooney-Rivlin model have been applied: \( C_{00}=889.490 \text{ kPa}, C_{01}=-245.840 \text{ kPa}, C_{20}=-155.310 \text{ kPa}, C_{11}=93.786 \text{ kPa}, C_{02}=11.148 \text{ kPa} \). The bulk modulus was set to be 2.5 GPa, as commonly used for elastomers and rubber-like materials.

2.1. Modal analysis

For both models and two load cases considered in the study, dynamic parameters, such as modes of free vibrations and corresponding natural frequencies, were first determined. The
estimated values of natural frequencies corresponding to the 1st modes of free vibrations for the empty as well as for the occupied structure are summarized in Tab. 1.

Table 1: Natural frequencies for empty and occupied temporary steel grandstand (1st modes of free vibrations in Y direction)

<table>
<thead>
<tr>
<th>Model</th>
<th>Natural frequency [Hz]</th>
<th>Empty structure</th>
<th>Occupied structure</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model 1</td>
<td>8.258</td>
<td>4.046</td>
<td>3.494</td>
</tr>
<tr>
<td>Model 2</td>
<td>8.224</td>
<td>3.494</td>
<td>3.494</td>
</tr>
</tbody>
</table>

Different changes of natural frequencies for Model 1 and Model 2 of grandstand equipped with typical diagonal stiffener and polymer element are connected with a relatively high stiffness of the structure itself.

2.2. Dynamic analysis

The second stage of the numerical study was devoted to dynamic transient analysis. In the analysis, the dynamic load has been assumed to be consisted of synchronous repetitive impacts, as expressed by Fourier series (see [1]). The full load was applied in the vertical direction together with the additional load of 6% in its value acting in the horizontal direction (according to Polish Standards).

The peak values of accelerations and displacements in Y direction of the grandstand under human-induced excitation due to jumping were determined (at the level of the highest steel platform) and summarized in Tab. 2. It can be seen from the table that the peak value of 0.46g (where g is the acceleration of gravity) was obtained for the analysed grandstand with the typical stiffener member, while the value of only 0.17g has been reached in the case of the structure equipped with polymer element. The first value can be considered unacceptable since the lowest recommended value of acceleration to avoid spectators panic is equal to 0.35g [1].

Table 2: Peak horizontal accelerations and displacements estimated for the grandstand (Y direction)

<table>
<thead>
<tr>
<th></th>
<th>Acceleration [m/s²]</th>
<th>Displacement [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model 1</td>
<td>4.482</td>
<td>4.020</td>
</tr>
<tr>
<td>Model 2</td>
<td>1.649</td>
<td>3.931</td>
</tr>
</tbody>
</table>

3. Conclusions

The results of the study clearly show that the response of a temporary steel grandstand equipped with the diagonal polymer element as well as with the diagonal stiffener is substantially different. The application of the polymer element leads to substantial reduction (by as much as 63%) in the level of computed accelerations, although the peak displacement values are nearly the same. This substantial decrease in the level of accelerations is caused by the enhanced damping properties of polymer.

Further detailed experimental study is planned to in a key issue of verifying damping properties.

References


392
Calibration of a parallel kinematic machine tool utilizing a MEMS inertial measurement unit

Björn Lünemann¹, Elmar Wings², Edward Chlebus³

¹, ² Faculty of Technology, Dept. Mechanical Engineering, University of Applied Science Emden/Leer
Constantiaplatz 4, 26723 Emden, Germany
e-mail: bjoern.luenemann@hs-emden-leer.de ¹, elmar.wings@hs-emden-leer.de ²
³ Faculty of Mechanical Engineering, Wroclaw University of Technology
5 Lukasiewicza Street, 50-371 Wroclaw, Poland
e-mail: edward.chlebus@pwr.wroc.pl

Abstract

The paper at hand illustrates a new concept for machine tool calibration utilizing a MEMS inertial measurement unit. Inertial measurement units (IMU) are devices for determining orientation and position that were initially developed for applications in the aerospace industry. Due to the triumphal procession of MEMS fabrication, the former complex and costly devices have become a popular solution for determining the orientation in recent years. Nowadays IMU’s can be found even in consumer applications like e.g. navigation and mobile devices. Unfortunately, the accuracy and reliability of the devices was not sufficient machine tool calibration. However, this situation might change with the newest incarnation of MEMS IMUs. At the University of Applied Science Emden/Leer a new calibration system is under development that incorporates a state of the art industrial grade MEMS IMU. Within the course of the paper we illustrate our concept for integration of the MEMS-IMU into the control system environment of the machine tool.

Keywords: machine tools, calibration, inertial measurement unit, MEMS

1. Introduction

Independent of the kinematic design of a machine tool, the industrial application requires an efficient calibration method for the investigation of its accuracy. The accuracy of a machine tool is primarily defined through geometrical and dynamical machine properties. Any deviation of the tool center point from the programmed position will be represented as geometric deviation in the workpiece. Therefore, it is necessary to measure deviations of the machine movements and eventually compensate them.

1.1. Calibration process

The first step in the calibration process is the establishment of a kinematic model of the machine tool. The goal within this step is to find an equation describing the pose of the machine tool dependent on the parameters that describe the geometry and the actual orientation of the joints. The following Eqn (1) from [6] depicts this dependency:

\[ P = f(\eta, \Theta) \]  

<table>
<thead>
<tr>
<th>P: Pose in task space</th>
<th>( \eta ): Vector of constants describing the geometry of the manipulator</th>
<th>( \Theta ): Vector of joint displacements for any particular pose</th>
</tr>
</thead>
</table>

A machine tool will most definitely be affected by errors, induced during the manufacturing and assembly. These might be deviations in the length of links or joint orientations. Hence, the kinematic model developed in the first step must be extended. The task is to find the error parameters that have a significant impact on the position and orientation of the TCP. In other words: Assess the impact of changes in uncertain parameter values on the output of the model. For the extended kinematic model only the parameters with major impact will be used. Calibration requires an accurate and reliable method for measuring the end effector pose or a subset of the pose in different joint orientations. Furthermore, a strategy for carrying out the measurements must be defined. The observation strategy should be designed in such a way, that the required number of measurements can be reduced to minimum.

The identification step is the main challenge within the calibration process. The purpose of the identification step is to choose the vector of model coefficients \( \eta \) that will minimize \( P_{pi} \) in some sense for the set of measured poses [6].

\[ \delta P_i = P_{mi} - P_{pi} \]  

\( P_{mi} \): Measured pose at \( i \)-th joint displacement \( P_{pi} \): Pose predicted by kinematic model

\[ \delta P_i = P_{mi} - f(\eta, \Theta) \]  

The final steps are the compilation of a calibration protocol and the adaptation of the kinematic model, taken into account the parameters found in the identification step of the calibration process.

2. Calibration of a parallel kinematic machine tool utilizing an IMU

The choice of the measurement technology and strategy is a crucial part within the calibration process. In recent years, some research has been conducted, that investigated the utilization of MEMS inertial sensors for calibration. Tabatabei [10] used MEMS acceleration sensors for calibration of a planar kinematic. Guanglong [11] investigated the utilization of a MEMS IMU for calibration of a serial robot. Gao, et. al. [2] investigated the utilization of an inertial measurement unit for a parallel kinematic machine tool.
2.1. Inertial measurement unit

An inertial measurement unit (IMU) is a device that can be used to determine the orientation and position of an object in space. For this purpose it contains two triplets of sensors. One of these consists of three angular rate sensors, while the other consists of three acceleration sensors. In both triplets the sensors are oriented orthogonal to each other. If the initial position and velocity of an object are known, the position can be calculated by a simple double integration of the acceleration over time. This basic principle is also used for IMU-navigation. However, the method for integrating the data is complex.

3. Concept for the IMU-based calibration system

Figure 1 illustrates the concept for a novel calibration system currently under development at the University of Applied Science Emden-Leer. The system makes use of the open architecture of the TwinCAT 3 environment. The inertial measurement unit transfers measurement data in real time to the control system. The core of the IMU algorithm is an extended Kalman filter. This well known algorithm has been adopted to the specific needs of the application at hand. The main task of the kalman filter is to calculate the best estimate of the position and orientation. Both, data from the IMU as well as nominal values delivered by the CNC are fed into the algorithm, taking into account the noise in the IMU data and in the encoder values of the machine tool itself. Finally the IMU algorithm provides data for compensation of deviations of the TCP.

References


Modelling of the vibration reduction system used for protection of working machine operators

Igor Maciejewski¹, Tomasz Krzyżyński²
¹,²Department of Mechatronics and Applied Mechanics, Faculty of Technology and Education, Koszalin University of Technology
Sniaideckich Str. 2, Koszalin, 75453 PL
e-mail: igor.maciejewski@tu.koszalin.pl¹, tomasz.krzyzynski@tu.koszalin.pl²

Abstract

In the paper the generalised model is proposed to determine the basic characteristics of non-linear visco-elastic elements used in typical vibration reduction systems. The proposed model takes into account the resonances of human body parts and organs in order to minimise the harmful vibrations affecting the machine operators during their work. Such a mathematical description of the vibration reduction systems determines the ability to use the presented model in a wide range of applications.

Keywords: vibration reduction, whole-body vibration, generalised model

1. Introduction

The fundamental method of evaluating the effectiveness of vibration reduction system is to perform an experiment in the laboratory. The tested system should be excited by the signals that are representative of the different type of working machines [?]. Based on measured signals, the vibro-isolation criteria of the tested system can be calculated. However, in many cases the test execution is difficult due to the technological restrictions and the high cost of the experiment. The duration of the test is also an important aspect, especially when the test must be performed repeatedly for different design parameters of the system [?]. Taking into account the complexity of the research process, the authors of the paper recommend to carry out a simulation experiment based on a generalised mathematical model of vibration reduction system, is shown in Fig. ??.

![Concept model of the vibro-isolation process](image1)

![Generalised model of the non-linear vibration reduction system](image2)

There are dozens of human body models presented in the modern literature [?, ?]. Usually, there are multi-degree of freedom lumped parameter models that consider seating and standing position. In this paper a generalisation of the well-known human body models is proposed that allows to use various biomechanical structure in the modelled vibration reduction system. An innovation of the model consists in a configurable mechanical system that can be used for the purpose of selecting the vibro-isolation properties of various suspension systems.

2. Generalised model of the system

In order to analyze the dynamics of such system (Fig. ??), models of the vibration reduction system and the isolated body have to be created. The bio-mechanical models are created in order to verify the vibration impact on individual parts and organs of the human body. Using these models, the vibration amplitudes and frequencies can be determined without the necessity of an experimental investigation. This kind of research may not be always reliable due to subjective feelings of the individuals. According to the paper [?], the significant discrepancies of measurement results occurred in the past, therefore the modelling studies are carried out at present.

The general model of vibration reduction system is presented in Fig. ?? The suspended body is isolated against harmful vibrations in three orthogonal directions: longitudinal x, lateral y and vertical z. The passive visco-elastic elements as well as the
active force actuators are used in order to minimise human exposure to a whole-body vibration. Rotation vibrations around each axis of the Cartesian coordinate system \((x, y, z)\) are neglected in this simplified model. The ongoing research \([?]\) indicates that the exposure of workers to the risks arising from vibration are evaluated for translational axes, therefore in the following paper the vibro-isolation properties are discussed a specific direction.

A set of three independent equations of motion is formulated in the matrix form:

\[
\mathbf{M}_i \ddot{\mathbf{q}}_i + \mathbf{D}_i \dot{\mathbf{q}}_i + \mathbf{C}_i \mathbf{q}_i = \mathbf{F}_{ai} + \mathbf{F}_{si}, \quad i = x, y, z \quad (1)
\]

where: \(\mathbf{q}_i\) is the displacement vector of isolated body, \(\mathbf{M}_i, \mathbf{D}_i, \mathbf{C}_i\) are the inertia, damping and stiffness matrices, \(\mathbf{F}_{ai}\) and \(\mathbf{F}_{si}\) are the vectors of exciting and active forces describing the non-linear vibration isolator.

The \(n\)-element vector represents the movement of elements contained in the bio-mechanical model of a human body:

\[
\mathbf{q}_i = [q_{1i}, q_{2i}, \ldots, q_{ni}]^T, \quad i = x, y, z \quad (2)
\]

where: \(q_{1i}, q_{2i}, \ldots, q_{ni}\) are the displacements of different parts of the human body.

The \(n\)-element vectors of exciting and active forces are given by the following expressions:

\[
\mathbf{F}_{ai} = [F_{ai, 0}, 0, 0]^T, \quad i = x, y, z \quad (3)
\]

\[
\mathbf{F}_{si} = [F_{si, 0}, 0, 0]^T, \quad i = x, y, z \quad (4)
\]

The particular non-linear exciting forces \(F_{ai}\) can be described in a general form as follows:

\[
F_{ai} = \sum_{j=1}^{k} F_{aij}(q_{1i} - \dot{q}_{ai}) + \sum_{j=1}^{h} F_{aij}(q_{1i} - \ddot{q}_{ai}), \quad i = x, y, z \quad (5)
\]

where: \(F_{aij}(q_{1i} - \dot{q}_{ai})\) are the non-linear functions including force characteristics of the conservative elements as a function of the system relative displacement \(q_{1i} - \dot{q}_{ai}\), \(F_{aij}(q_{1i} - \ddot{q}_{ai})\) are the non-linear functions including force characteristics of the dissipative elements as a function of the system relative velocity \(\dot{q}_{1i} - \ddot{q}_{ai}\). The input displacement \(\dot{q}_{ai}\) and velocity \(\ddot{q}_{ai}\) are modelled as excitation signals that are generated for the specific direction of vibration exposure \(x, y, z\).

The non-linear active forces can be described as a function of the input signals \(u_{i}\) and the system relative displacement \(q_{1i} - q_{ai}\) or velocity \(\dot{q}_{1i} - \dot{q}_{ai}\) as follows:

\[
F_{ai} = \sum_{j=1}^{k} F_{aij}(u_{i}, q_{1i} - q_{ai}, \dot{q}_{1i} - \dot{q}_{ai}), \quad i = x, y, z \quad (6)
\]

where: \(F_{aij}\) defines dynamic characteristics of the actuators.

A generalised structure of the human body model proposed within the framework of the paper is shown in Fig. ?? This human body is modelled as a discrete mechanical system with many degrees of freedom and its particular bodies are combined using visco-elastic elements.

In the paper, the basic structure of a vibration reduction system is employed to formulate the generalised model of human body exposure. Such a model can be used to describe the general dynamic behaviour of many vibration reduction systems. However, their essential force characteristics \((F_{cij}, F_{dij}, F_{aij})\) have to be evaluated for the elements, e.g., mechanical, pneumatic, hydraulic, etc., applied in the suspension systems \([?]\).

3. Conclusions

The presented model provides the basis using a mechanical analogue of the human body. Such a mechanical system should assist an appropriate selection of the vibro-isolating properties in view of the conflicted requirements for modern vibration reduction systems. Such a selection of the dynamic characteristics can be conducted for the vibration isolators of various designs that are activated using the excitation signals representing the work of diverse types of the machinery.

References


Identification of defect factors for a road bridge made of pre-stressed concrete on the basis of static strength analysis

Arkadiusz Madaj¹, Wojciech Siekierski²
¹,² Faculty of Building and Environmental Engineering, Poznań University of Technology
ul. Piotrowo 5, 61-138 Poznań, Poland
e-mail: arkadiusz.madaj@put.poznan.pl¹, wojciech.siekierski@put.poznan.pl²

Abstract

The paper concerns identification of defect factors for a road bridge made of pre-stressed concrete through static strength analysis. It shows that the reasons for observed defects are faulty parameters of prestressing (tendon layout, prestressing force and, probably, underestimation of prestressing force losses). It also shows that the defect factors could be identified on the basis of bridge profile line measurement taken after prestressing.

Keywords: pre-stressed concrete bridge, defect factor identification, static strength analysis

1. Introduction

Identification of defect factors for a given structure is necessary to choose the method of its repair. One of the identification techniques is analysis of internal force and displacement distribution against defect results. The paper shows its application in the case of a road bridge.

2. Characteristics of structure and its defects

The bridge in question is a twin girder triple-span structure made of pre-stressed post-tensioned concrete. All spans are equal. The bridge cross-section changes from twin T-beam girders to single-cell box girder in the vicinity of piers (Fig. 1). It results in variation of neutral axis level and flexural stiffness. It influences normal stress distribution due to prestressing.

![Figure 1: Bridge cross-section: at span (left), at pier (right)](image)

Figure 2: Layout of pre-stressing tendons and location of observed cracks (see text for tendon symbols description)

The bridge concrete superstructure was completed on scaffolds and then pre-stressed with continuous tendons running along the whole bridge length, short tendons in terminal spans and tendons over piers (in two groups – longer and shorter) – Fig. 2. The available data on prestressing include: tendon type (the cross-section) and nominal, initial prestressing force for each tendon. Prestress technology data, including sequence of prestressing, is not available.

Couple of years after bridge completion cracks of both girders occurred near piers, mainly in the middle span (Fig. 2). The cracks extend up to 1/3 of girder depth (in average). The cracks are about 0.3 mm wide at the girder bottom edge. Permanent span deflections were also recorded. They were reported just after formwork dismantling. During service period the deflections have been stable (Fig. 3). The designed bridge profile line was not achieved what can be graded as structural fault, similarly to cracks.

3. Hypothesis concerning defect factors

Based on in-situ bridge inspection and analysis of available design and erection documentation the hypothesis was put forward that the observed defects (including permanent bridge span deflection) is a result of faulty design of prestressing, particularly underestimation of prestressing force losses, especially for continuous tendons running along the whole bridge length. Observed results of the fault – single cracks of relatively substantial width – resulted from insufficient reinforcement near bottom edge of girders in the vicinity of intermediate supports.

The hypothesis stated above was verified based on internal force and deformation distribution. Dead loads, live loads and prestressing, as external loading, were considered.

4. Computational model

Finite element method, implemented in the Robot package, was applied to analyze the bridge. Structure and member dimensions were assumed based on design documentation. The
structure was modelled with 2-node beam elements. Longitudinal beam elements represent main girders while transverse beam elements – deck slab. Its thickness variations are taken into account as well as variations of main girder cross-section and neutral axis level near piers.

5. The method of hypothesis verification

5.1. Criteria of defects identification

Identification of defects requires setting the criteria that allow to make a judgement that the analysed structure has reached its ultimate capacity. In the analysed case there are two such criteria: cracking and deflection.

Tensile stress at the girder edge fibre greater than tensile strength of concrete (ultimate strain) was assumed as cracking criterion. Since reinforcement coefficient at critical cross-sections of main girders does not exceed 0.04% the effect of tension stiffening does not occur and it was justified to assume that the member behaves as if it was made of concrete (strain energy concentrates in a single cross-section resulting in a single, relatively wide crack).

Identification of prestressing effectiveness can be done by comparing designed vertical displacements of a structure after prestressing with the recorded ones. Recorded displacements allow for estimation of actual values of prestressing forces.

5.2. Analysis of normal stress distribution

Since cracking and span deflection are observed in absence of live loads the static strength analysis was carried out for a combination of dead loads and prestressing. Based on the hypothesis that the main reason for the observed defects was underestimation of prestressing force losses analyses were carried out for various combinations of prestressing forces.

Regarding only time-dependent prestressing force losses (coefficient of prestressing force losses $\eta=0.85$, equal for all tendons) generates very small compressive normal stresses at the bottom fibre at the cracked cross-sections. Since it happens for constant prestressing force losses it shows that it is the effect of prestressing tendon layout and the variation of main girder cross-section – variation of neutral axis location where twin T-girder cross-section changes into single-cell box girder cross-section.

A similar analysis was carried out for prestressing forces accounting for immediate and time-dependent losses. The following total coefficients of prestressing force losses were assumed (according to the assumed tensioning sequence): $\eta=0.85 / 0.75 / 0.60 / 0.50$ (three changes between anchorage and beam symmetry plane) for continuous tendons (no.1÷3), $\eta=0.75$ for longer tendons over piers (no.7÷10), $\eta=0.80$ for short tendons in terminal spans (no.4÷6), $\eta=0.80$ for shorter tendons over piers (no.11÷13).

Computational results show (Fig.4) that the location of minimum compressive stresses coincide with location of observed cracks in the middle span (Fig.2). These are the first cracks that after years have the biggest width and length.

The stress diagram in Fig.4 does not include component due to concrete shrinkage that exceeds 1 MPa at the bottom fibre. The observed cracks were initiated by service loads that were not included in the analysis due to unknown intensity and spectrum.

5.3. Analysis of span deflection distribution

Vertical displacements of structure were also analysed in a way similar to stress analysis. While considering only time-dependent losses the prestressing causes decrease of deflection in the middle span, however not big enough to compensated initial deflection of formwork – about 1 cm instead of over 6 cm. Analysis accounting for immediate and time-dependent losses (under assumption similar to those for stress analysis) shows that prestressing increases deflection in the middle span beyond those of formwork by about 1 cm and lifts terminal spans – Fig. 5.

6. Conclusion

The presented analysis shows that based on static strength computations it is possible to identify causes of observed defects of structure (a bridge in the example). In the described case the defects occurred probably due to underestimation of prestressing force losses. The first signal of faulty assessment of influence of prestressing on internal forces was insufficient lift of bridge spans after prestressing. It was proved afterwards by occurrence of cracking at the cross sections that were identified as critical in the view of results of static strength analysis accounting for prestressing force losses. The analysis also shows that it was possible to foresee cracking immediately after structure completion (formwork dismantling after prestressing).

References


Reference FEM model for SHM system of cable-stayed bridge in Rzeszów

Aleksandra Mariak1, Mikołaj Miśkiewicz2, Błażej Meronk1, Krzysztof Wilde4

1,2,3,4 Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: alemaria@pg.gda.pl 1, mmisk@pg.gda.pl 2, blazej.meronk@pg.gda.pl 3, wild@pg.gda.pl 4

Abstract

The paper presents the reference model for structural health monitoring system (SHM) of a cable-stayed bridge located in Rzeszów. The SHM system should be reliable, with a clear, simple design to ensure full functionality for the evaluation of technical condition of the bridge during its entire life cycle. The bridge SHM system is equipped with additional, innovative ultrasonic system for monitoring of material degradation of concrete.

Keywords: cable-stayed bridge, health monitoring, finite element model, ultrasonic diagnostics

1. Introduction

The SHM system is a kind of continuous non-destructive diagnostic system. Sensors and actuators of the SHM system are permanently mounted on the object or even built into it. The aim of SHM system application is to determine technical condition of an engineering object by means of objective methods and to increase structure durability, reliability and effectiveness. Information on the object technical state is a base for detection of the existence of damage, detection of damage location, identification of damage type and extent as well as prediction of damage development. The above tasks facilitate the maintenance decisions at each phase of the object life.

The advanced SHM systems use theoretical and numerical models for precise definition of object’s technical state. One of a possible solution for the advanced SHM system is application of reference FEM models [1]. The block diagram of this solution is shown in Fig. 1. The diagnosis is obtained by means of analysis of changes between the results from the numerical models in reference state, current state and damaged state with respect to data collected from the SHM system sensors.

2. Description of the bridge of SHM system

2.1. Research object

The bridge is a 5-span cable-stayed structure with length 3 x 30,00 + 150,00 + 240,00 m [2]. The fan type of cables have been designed. Total high of a bridge tower equals 108,50 m.

The center spacing of cable anchorage in the tower varies in the range from 1,70 m in the top to 2,40 m in the bottom. The inner part of the tower was intended for communication to the tower top. The superstructure of the bridge is concrete deck slab with steel box girder jointed by floor beam and hung to tower structures with cable spaced of 12,00 m. The bridge is designed for class load A according to PN-85/S-10030 and STANAG 2021 class 150 according to Dz. U. Nr. 63 [3].

2.2. SHM system

The main hardware element of the SHM system is a central management unit, which controls the operation of the system, collects measurement data, performs numerical simulations and a relevant analysis [3]. The monitoring system consists of three functional modules: a measurement, expert and notification parts. The first module collects and stores data from sensors and measuring devices. The expert module analyses the environmental impacts and response structure in time. It also includes a segment numerical simulations and the algorithm for generating alert. Notification module informs about exceeding safety factor. A view of the bridge and SHM system is shown in Figure 2.

The measurement module consists of the following segments:

- acceleration measurements: tower (1 pc., 2 measurement channels - top of the tower), cables (14 pc., 84 measurement channels), deck (4 pc., 24 measurement channels - section B-B, D-D);
- force in cable measurement: (direct measurements of changes in tension forces in 2 reference cables);
- angle measurements: tower rotate angle (1 pc., 2 measurement channels - top of the tower), deck rotate angle (6 pc., 12 measurement channels - section C-C, D-D, E-E);
- strain measurements: deformations of deck (24 pc. - section C-C, D-D), temperature measurement in deformation point;
- normal force in cables: (12 pc., 72 measurement channels);
- meteorological: temperature, wind direction and speed (1 pc. - top of the tower);
- ultrasonic measurement: diagnostics of concrete (1 pc. - bottom of the tower).

Figure 1: The block diagram of SHM system with a FEM reference model.

Figure 2.
3. Reference FEM model of the bridge

A detailed three-dimensional (3D) FE model of the as-built bridge has been developed in Sofistik software. The mesh of the FE model and numbers of cables are shown in Figure 3. The concrete deck is represented by quad elements, the girder and tower with beam element and cables applying truss element. The number of degrees of freedom equals 176353.

4. Selected results

The aim of the numerical analysis is to determine the variability of parameters selected to describe technical condition of the bridge. Changes of normal forces in cables under live load (vehicle $K$ and load $q$) were considered. In addition, a damage scenario associated with loss of tension force in cable no. 31 and 32 (Fig. 3) has been taken into account. Three loadcase combinations are defined: comb. I - dead load + equipment, comb. II - dead load + equipment + $K$ + $q$ (Fig. 4), comb. III - dead load + equipment + $K$ + $q$ in the case of breaking the cable number 31 and 32. The selected results of changes of normal forces are summarized in Table 1. Due to damage of two cables the force in cable no. 30 increases by 925 kN and the force in cable no. 1 decreases by 634 kN.

Table 1: Numerical results

<table>
<thead>
<tr>
<th>No. cable</th>
<th>Normal forces $\Delta F$ [kN]</th>
<th>$\Delta F$ [kN]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>+1890</td>
<td>-634</td>
</tr>
<tr>
<td>2</td>
<td>+576</td>
<td>-151</td>
</tr>
<tr>
<td>3</td>
<td>+1166</td>
<td>-58</td>
</tr>
<tr>
<td>29</td>
<td>+1266</td>
<td>-642</td>
</tr>
<tr>
<td>30</td>
<td>+1103</td>
<td>+925</td>
</tr>
<tr>
<td>31</td>
<td>+457</td>
<td>broken</td>
</tr>
<tr>
<td>32</td>
<td>+980</td>
<td>broken</td>
</tr>
</tbody>
</table>

5. Conclusions

As part of an ongoing long-term monitoring research project, a finite element model has been developed to monitor the behaviour of the road bridge over the Wisłok River in Rzeszów. Based on on-line measurements, the Expert module equipped with the FEM reference model will trace the changes in the force distribution in the object structural elements. The monitoring of these changes allows to estimate the current technical condition of the bridge and to share information with a team responsible for a bridge management.

References


Calculated and measured dynamic properties of the FRP composite beam

Barbara Markiewicz¹, Maciej Kulpa², Leonard Ziemiański³
¹,²,³ Faculty of Civil and Environmental Engineering and Architecture, Rzeszów University of Technology
Powstańców Warszawy 6, 35-939 Rzeszów, Poland
e-mail: bmarkiewicz@prz.edu.pl ¹, kulpa@prz.edu.pl ², ziele@prz.edu.pl ³

Abstract

The paper presents dynamic research carried out on a beam made of a fibre-reinforced polymer (FRP). The beam is a tested fragment of a girder in simply supported footbridges. The experimental and operational modal analysis have been performed for several variants of excitation and of the measured values. Frequencies, mode shapes and modal damping have been obtained. The behaviour of the composite beam was described by various models, starting with a simple prismatic beam with mean material properties for which the frequency has been calculated using the closed-form solutions. Then the additional features and applied different theories have been taken into account. The detailed shell FEM model was also created. The results for all the numerical models were presented as frequencies of particular modes and compared with those obtained for physical model. The purpose of the paper is to investigate model parameters describing the model to significantly affect the results in the modal analysis.

Keywords: modal analysis, FEM modelling, beam, footbridge, FRP composites

1. Introduction

Nowadays the computer programs allow to achieve a very accurate representation of the real structure by a numerical model. However, the more sophisticated the model is, the more difficult is to recognise and to find any potential modelling errors. Therefore, only those system properties and phenomena should be taken that are essential for conducted analysis those of no significant effect on the final results should be excluded. Thus, the numerical model can be easily modified and adjusted to a real, physical model.

2. The tested beam

<table>
<thead>
<tr>
<th>Material</th>
<th>Young’s modulus E (GPa)</th>
<th>Poisson’s ratio νxy</th>
<th>Shear modulus G (GPa)</th>
<th>Specific weight ρ (kN/m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Glass FRP</td>
<td>24,0</td>
<td>0,14</td>
<td>4,37</td>
<td>21,5</td>
</tr>
<tr>
<td>Carbon FRP</td>
<td>125</td>
<td>0,31</td>
<td>4,00</td>
<td>15,7</td>
</tr>
</tbody>
</table>

The paper presents dynamic models of a composite beam, which acts as a girder in simply supported footbridges. The tested beam off total length 13.5 m and span equal 12 m consists of two basic elements: a box girder and deck (Fig.1 a). The top and bottom chords consist of alternating laminates of glass and carbon FRP, and act as main load-bearing elements in the cross-section. Box beam webs are composites of two outer layers of glass FRP and filling of 15 mm of foam. The box girder is stiffened by similarly formed, composite diaphragms and external ribs (Fig.1 b). The parameters of the composite layers were presented in Table 1.

3. Beam model

The material is assumed to have mean material constants of individual components, based on the percentage of material in cross-sectional area. In the first case has been considered a mean modulus, was considered based only on moduli Ex of the layers, while the second variant was based on the mean of moduli: Ex and Ey (called Exy). For segments of the beam without stiffening elements, basing on information in Table 1, Ex=57.3 GPa and Exy=38 GPa. The segments stiffened with diaphragms and ribs were assigned material constants: Ex=41.2 GPa and Exy=31.2 GPa.

The characteristics for the actual cross-section were calculated as for a homogeneous material, as the second variant, weighted characteristics were created of a cross section of two materials for two options (Ex and Exy).

The calculated weight of the beam has been 118 kg/m, and for segments with additional stiffening elements: 290 kg/m.

A sequence of models begins with a prismatic beam with a continuous distribution of mass with the frequency calculated using the closed-form solutions. Euler-Bernoulli beam theory is followed by Timoshenko theory, which takes into account the influence of shear deformation and rotational inertia effects [2]. The ADINA step by step includes: overhangs, stiffening by ribs and the load caused by concrete fillings in supports area: 2850kg over the left one and 730kg over the right.
The results for all the models were represented by frequencies in [2]. The calculations were performed for two cases: with the rotation around the x axis restrained in single support or in both supports. Due to the model including additional masses of concrete the scheme of the support is no longer symmetrical, and these cases were regarded two different models: with restrained x-rotation on the left support or on the right.

The frequencies presented in Table 2 were obtained for model made of homogeneous material with Exy modulus, including all mentioned elements and with restrained x-rotation on the left support.

4. FEM model

One of the goals of the research project was to build and validate the detailed FEM model of the girder. To follow the numerical analysis the finite element modelling environment SOFISTIK 2010 was applied. The four-node composite plate finite elements (quad) have been mainly used for girder discretization. Only the concrete filling in the support region was modelled using brick elements. The entire beam model consisted of 4744 nodes with 7390 plate and 600 brick finite elements [1]. The visualization of the numerical model is shown in Fig. 2.

![FEM model of the beam](image)

Figure 2: FEM model of the beam

The material properties were obtained from the testing (Table 1). The orthotropic properties of the laminates e. i. different characteristics in different directions have been also considered. Thanks to software code possibilities, the exact layer structure of the laminates was discretized into 10 various layers described as plate finite elements. Thus the exact material properties was considered in the girder model with a combination of three basic materials: carbon and glass laminates and core foam.

The shell model was supported on the entire width of the beam, without restrained x-rotation.

5. Experimental measurement

Dynamic properties of the composite girder were studied in experimental and operational modal analysis. The following dynamic parameters of the girder was obtained: frequencies, mode shapes and modal damping. The excitation was generated at one point (by modal hammer, modal shaker or by kinematic excitation), and the response, in the form of accelerations, has been measured at multiple other points. Such an approach is defined as SIMO (single input - multiple output). In the studies used several types of excitation were used: impact force, random noise excitation, harmonic and kinematic.

The experiment was performed 40 measurement cycles, for different variants of excitation and for different variants of the measured values. The acceleration signals were measured by accelerometers, in the form of frequency response functions (FRF), power spectra and the coherence function. During all measurements the FRF functions were recorded, and using the Polymax estimation method, the stability diagrams created. The result were determined the modal parameters of the tested composite beam. Analyzing the obtained measurement results, it can be concluded that the tested beam has had a high modal density. The contained in a range 0-120 Hz number of identified modal frequencies is 30. Received, during experimental and operational modal analysis, values of parameters have low standard deviation, regardless of the type of excitation. The measurement results were presented in Table 2, and compared with the results obtained from the FEM analysis and beam model analysis.

6. Conclusions

The paper describes a sequence of simple dynamic models of the beam followed by complex shell FEM model. Parameters that may affect the frequencies were considered. The frequencies, mostly corresponding with the frequencies of the shell were presented. The results for beam model are corresponding with those obtained for detailed FEM model in the range of low frequencies.

References


<table>
<thead>
<tr>
<th>ID</th>
<th>Measured frequencies [Hz]</th>
<th>Mode shape</th>
<th>Frequencies of FEM model [Hz]</th>
<th>Mode shape</th>
<th>Frequencies of beam model [Hz]</th>
<th>Mode shape</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9.7</td>
<td>bending - xy</td>
<td>16.3</td>
<td>bending - xz</td>
<td>16.5</td>
<td>bending - xz</td>
</tr>
<tr>
<td>2</td>
<td>16.1</td>
<td>bending - xz</td>
<td>17.8</td>
<td>bending - xy</td>
<td>18.4</td>
<td>bending - xy</td>
</tr>
<tr>
<td>3</td>
<td>20.8</td>
<td>torsional</td>
<td>26.8</td>
<td>torsional</td>
<td>21.0</td>
<td>torsional</td>
</tr>
<tr>
<td>4</td>
<td>26.2</td>
<td>torsional</td>
<td>30.2</td>
<td>torsional</td>
<td>41.3</td>
<td>bending - xz</td>
</tr>
<tr>
<td>5</td>
<td>31.0</td>
<td>torsional</td>
<td>36.5</td>
<td>bending - xz</td>
<td>39.1</td>
<td>longitudinal</td>
</tr>
<tr>
<td>6</td>
<td>33.9</td>
<td>bending - diagonal</td>
<td>43.5</td>
<td>local</td>
<td>72.6</td>
<td>torsional</td>
</tr>
<tr>
<td>7</td>
<td>49.2</td>
<td>torsional</td>
<td>45.3</td>
<td>local</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>8</td>
<td>51.6</td>
<td>bending - xy</td>
<td>48.4</td>
<td>local</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>9</td>
<td>56.6</td>
<td>torsional</td>
<td>50.4</td>
<td>local</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>10</td>
<td>62.7</td>
<td>bending - xy</td>
<td>53.2</td>
<td>local</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>
Numerically based quantification of internal forces generated in steel sway frame structures with flexible end-plate joints, exposed to fire

Mariusz Maślak¹, Michał Pazdanowski², Małgorzata Snela³

¹² Faculty of Civil Engineering, Cracow University of Technology
Warszawska 24, 31-155 Cracow, Poland
e-mail: mmaslak@pk.edu.pl¹, plpazdan@cyfronet.krakow.pl²
³ Faculty of Civil Engineering and Architecture, Lublin University of Technology
Nadbystrzycka 38D, 20-618 Lublin, Poland
e-mail: m.snela@pollub.pl

Abstract

The distribution of internal forces generated in the steel frame subjected to fire is determined by the flexibility of joints connecting the structural members. Such flexibility, increasing with growing member temperature, may be analysed by means of a generalized component method introducing a special finite element at each joint, with properties corresponding to structural response of particular joint components. For small size frames analogous analysis may be performed with 3D finite elements applied to discretize the whole considered structure. Thus on a small scale a more precise model may be used to calibrate and verify the parameters of a simplified model, later to be applied to analyze more complex structural systems. Identification of behaviour of a steel frame structure with a sway transversal frame and flexible beam-to-column joints subjected to fire is the objective of the paper. A beam element and 3D element FEM models of the frame are applied in the calculations.

Keywords: fire safety, end-plate joint, component method, model verification.

1. Introduction

The behaviour of steel frames subjected to fire is usually analysed using beam elements applied to model the structural components (beams and columns). The generalized component method [4,8] is recommended to determine the nonlinear relationships of the $M - \varphi$ type, accounting for the flexibility of joints increasing with the member temperature.

More reliable results should be available using more precise numerical model, based on 3D modelling of either a joint only, with remaining parts of the structure modelled the beam elements, or full 3D modelling of the whole analyzed frame [2]. The latter approach should yield the most accurate results, but it may be applied only for the structures of limited sizes [1].

Identification of the behaviour of a steel frame with sway transversal frames and flexible beam-to-column joints subjected to fire exposure, solved by means of the component method based and 3D analysis is the objective of this paper.

2. Mechanical model

In order to correctly estimate the fire resistance of a steel frame one should account for the joint rigidity decreasing with developing potential fire. This means that the joints designed nominally rigid for persistent design scenario, may become flexible during the fully developed fire. Introduction of this statement of this type to the traditional computational procedure, oriented on the evaluation of both the fire resistance and the stability of the analyzed frame, results in the need to specify a special nodal element, whose stiffness matrix $K'_c$ forms the equation:

$$ F = K'_c \cdot u $$

where $F$ denotes the vector of generalized internal forces generated in the considered joint, $u$ a vector of generalized displacements identified for this joint. Due to strong and multi-sourced nonlinearity of the analysis associated with thermal action of fire on a considered structure the equation (1) has to be solved iteratively, thus the matrix $K'_c$ becomes the tangent stiffness matrix $K'_c^T$. In such a case this equation is reduced to the following:

$$ \Delta F = K'_c^T \cdot \Delta u $$

where, respectively:

$$ \Delta F = \begin{bmatrix} \Delta N_{xj}, \Delta V_{yj}, \Delta V_{zj}, \Delta M_{qj}, \Delta M_{yj}, \Delta M_{zj} \end{bmatrix}, $$

$$ \Delta K'_c^T = \begin{bmatrix} \Delta N_{xj}, \Delta V_{yj}, \Delta V_{zj}, \Delta M_{qj}, \Delta M_{yj}, \Delta M_{zj} \\ \Delta N_{yj}, \Delta V_{yj}, \Delta V_{zj}, \Delta M_{qj}, \Delta M_{yj}, \Delta M_{zj} \\ \Delta N_{zj}, \Delta V_{yj}, \Delta V_{zj}, \Delta M_{qj}, \Delta M_{yj}, \Delta M_{zj} \end{bmatrix}, $$

$$ \Delta u = \begin{bmatrix} \Delta u_x, \Delta v_y, \Delta w_z, \Delta \phi_{xj}, \Delta \phi_{yj}, \Delta \phi_{zj} \end{bmatrix}, $$

$$ \Delta K'_c^T = \begin{bmatrix} \Delta u_x, \Delta v_y, \Delta w_z, \Delta \phi_{xj}, \Delta \phi_{yj}, \Delta \phi_{zj} \end{bmatrix}. $$

Identification of the components of the matrix $K'_c$ is usually performed based on the classical component method generalized to the case of fire. Thus the behaviour of the joint, especially its flexibility, is modelled with an appropriately selected set of springs, each spring corresponds to a conceptually isolated component of the considered joint. Each of such components is primarily assigned to the respective type of work, i.e. bending, compression, tension or shear. A diagram of a typical flexible end-plate joint connecting the beam with the column of a frame considered in the example is shown in Fig. 1., a listing the components determining its effective stiffness and an accompanying simplified mechanical model. The models of this type may be developed to a high extent, mainly to take into consideration the possible bending – shear interaction. This was shown by the Authors in [3].
Applying the formal model presented above for the persistent design scenario, one obtains a single nonlinear $M - \phi$ relationship between the bending joint moment $M$ and the resultant rotation $\phi$ of the beam with respect to the column. In order to account for the temperature of the joint components increasing in fire, a whole family of such relationships should be specified, each of them being a solution of an isothermal problem at a fixed steel temperature [7]. It was shown previously [4,8], that the $M - \phi$ curves relating to the fire scenario may be generated in a simplified way, through the transformation of an a priori known reference curve determined at room temperature. Such a method may, however, result in an overestimated joint rigidity, thus yielding an overestimated bearing capacity of the whole frame [5]. Opinion of the authors, a more accurate approach should be applied in this field, based on the generalization of a well-known component method. The first step is to identify potentially possible failure mechanisms of the joint components subjected to the foreseen fire, the second step to estimate the bearing capacity of each component related to the considered mechanism [9]. The bearing capacity is later transformed into the parameters $k_i$ which determine stiffness of the substitute springs modelling the behaviour of the $i$-th joint component in fire. This parameter is, in general, a nonlinear function of the steel temperature $\Theta$. Since the components of the stiffness matrix $K_i^*$ are based on the characteristics $k_i$ related to the previously identified joint components, they will change with increasing temperature of these components. However, the form of the matrix in the will not qualitatively change.

It is assumed that the fire is past the flashover point, and reached the phase of a fully developed fire. Thus it may be understood that the temperature of the exhaust gases in the whole building compartment is steady, but increasing with time. The analyzed frame, with support conditions and the detail of the considered joint are depicted in Fig. 2.

In the analyzed example the beam joins the column by an end-plate joint with four rows of bolts. Assuming increasing temperature of the steel elements, even subjected to heating, the changes in internal forces in both the column and beam of a frame are monitored. An observation of various global stability effects (such as push-out effect, pull-in effect, pullback effect, catenary effect), induced in the frame as a consequence of the substantial displacements in the deforming bearing structure [6] is essential. These effects are due to the redistribution of internal forces, typical for the case of fire. Identification of this redistribution pattern seems to be crucial for a credible estimation of fire safety for the entire structure. Behaviour of the sway frame with flexible joints on fire was preliminarily analysed by the authors [6]. Application of the nonlinear 3D FEM analysis should result in a much more precise evaluation of fire resistance of this kind of a frame.

It is the intent of the authors to use the relationships presented herein, to verify the results obtained with purely analytical algorithms, recommended for practical use so far.

References


Abstract

One of the crucial factors determining the effective burden is an appropriate selection of geometry of cut-holes. Furthermore, leaving empty (uncharged) blasting holes may lead to enlargement of the fractured zone, and thus – improvement of the mining faces blasting effectiveness by means of reducing the number of blasting holes and/or extending the burden. The aim of the paper is to present a theoretical basis and preliminary results of numerical analysis carried out to determine the fractured zone caused by detonation of explosives. The authors used Arbitrary Lagrangian–Eulerian formulation for modelling the pressure wave propagation and its interaction with the mining face.

Keywords: blasting technique, numerical modelling, blast simulation

1. Introduction

The most common method for excavating copper ore in Polish mining industry is a blasting technique, which involves drilling holes in the mining face. An actual number of holes, their relative position and even charging sequence are the factors influencing effectiveness of the process. Any optimization of this process is incredibly difficult since acquisition of any field data different than an amount of the factors influencing effectiveness of the process. Any optimization of this process is incredibly difficult since acquisition of any field data different than an amount of the factors influencing effectiveness of the process. Therefore, an idea to implement numerical methods to simulate mining face fragmentation emerged. In the article, the authors present an approach to simulate blast wave propagation in the mining face.

2. Numerical blast modelling

Out of many techniques of pressure wave propagation analysis, the authors focus on those involving discrete models. These methods are based on the Finite Element Method (FEM), which uses several algorithms for determination of dynamical loading of the structure, e.g. ConWep blast wave function, Smoothed Particle Hydrodynamics (SPH) method and Arbitrary Lagrangian-Eulerian formulation (ALE). In fact, the latter is adopted in the presented study. Such a choice is based on the previous study of several test models and the authors’ experience in this research area [4]. Investigations of the detonation process, pressure wave propagation and its interaction with structures can be found in many references [1,2,3].

The ALE procedure consists of two major steps: the classical Lagrangian step and the advection Eulerian step. The first step is carried out with the assumption that displacements of the nodes are very small in comparison to characteristics of the elements surrounding these nodes, e.g. dimensions. Moreover, in this procedure, a constant topology of the mesh is provided [5].

3. Materials models

Detonation of the explosive is simulated using a burn model based on the function that determines the degree of burn of an explosive in the given finite element. This function consists of two parts. The first one – \( F_1 \) describing burning with known predetermined speed \( DCJ \) and the second one – \( F_2 \), applicable where detonation velocity exceeds \( DCJ \). The bigger value of \( F_1 \) and \( F_2 \) is the final pressure multiplier resulting from the equation of state of the detonation products [6]:

\[
F = \begin{cases} 
\frac{2(t-t_i)DCJ}{3L} & \text{for } t > t_i \\
0 & \text{for } t \leq t_i
\end{cases}
\]

\[F_2 = \frac{1-V_CJ}{1-V_CJ} \tag{1}\]

where \( t - t_i = \frac{R_{det}}{DCJ} \) – moment of detonation, \( R_{det} \) – distance from the element centre to the point of detonation initiation, \( DCJ \) – detonation velocity, \( L \) – characteristic length of element, \( V_CJ \) – volume of Capman-Jouguet point.

The mining face is modelled using a material model proposed by Riedel et al. [7,8], known as RHT. It is a macroscale material model that incorporates features necessary for a correct dynamic strength description of concrete at impact relevant strain rates and pressures. The shear strength is described by means of three limit surfaces; an inelastic yield surface, a failure surface and a residual surface, all dependent on the pressure value. The post-yield and post-failure behaviours are characterized by strain hardening and damage, respectively. Furthermore, the pressure is governed by the Mie-Grüneisen equation of state together with a \( p-\alpha \) model (Eqn (2)) to describe the pore compaction hardening effects and thus give a realistic response in the high pressure regime.

\[
\alpha(t) = \max \left[ 1, \min \left( \alpha_0, \min \left( \frac{P_{\text{comp}} - P(t)}{P_{\text{comp}} - P_{\alpha}}, 1 + (\alpha_0 - 1) \left( \frac{P_{\text{comp}} - P(t)}{P_{\text{comp}} - P_{\alpha}} \right)^{\gamma} \right) \right) \right] \tag{2}
\]
In the above equation $\alpha$ represents porosity. The current pore crush pressure is given by the following formula [6]:

$$p_c = p_{\text{comp}} - \left( p_{\text{comp}} - p_{\text{el}} \right) \left( \frac{\alpha - 1}{\alpha_{0} - 1} \right)^{N},$$

(3)

where $p_{\text{comp}}$ – compaction pressure, $p_{\text{el}}$ – initial pore crush pressure, $N$ – porosity exponent.

4. Results

A number of different holes patterns are investigated for selecting the most effective one. Each of them have a prescribed specific detonation sequence. In Fig. 1, one of the simulated patterns is presented. Propagation of the blast wave in the mining face for one of the patterns is presented in Fig. 2

![Figure 1: A pattern of holes located at the center of the mining face, white holes denote the empty ones (not filled with an explosive).](image1)

![Figure 2: Propagation of the blast wave after the first blast (left) and after the second blasts (right)](image2)

![Figure 3: RHT model damage index after the first blast (left) and at the end of a blasts series (right)](image3)

In Fig. 3, a damage index of the RHT model after the first blast and after detonation in all holes is presented. The obtained results show that even a simplified numerical analysis can be very helpful in the selection of the most effective holes pattern. Therefore, the authors have undertaken further steps towards the acquisition of reliable RHT parameters and development of 3D models allowing better representation of the explosive burn process.

References


Numerical simulation of moving load effect on pavements

Jozef Melcer*
Faculty of Civil Engineering, University of Žilina
Univerzitná 8215/1, 010 26 Žilina, Slovak Republic
e-mail: jozef.melcer@fstav.uniza.sk

Abstract

The submitted paper is dedicated to the numerical simulation of moving load effect on road structures in time and frequency domains. The multi-body computing models of vehicles on various levels are introduces. The equations of motion are derived in the form of ordinary differential equations. The equations of motion are solved numerically in the environment of a program system MATLAB. The road unevenness as the main source of kinematical excitation of vehicles is modeled as the random rod profile by the use of the power spectral density functions. Especially the time histories of tire forces, frequency spectra and frequency response functions are the subject of interest.

Keywords: pavements, moving load, numerical simulation, tire forces, frequency spectra

1. Introduction

Dynamic effect of a moving load on transport structures can be followed in the literature from the year 1849, [3]. It was induced by the collapse of the Chester Rail Bridge in England in the year 1847, [4]. At the early stage the analytical methods were applied. The development of computers brings the revolution and qualitative jump in the development of solution. Presently the moving load effect on road structure can be simulated numerically in time and in frequency domain. The time history and frequency composition of tire forces from vehicles are the subject of interest of road designers.

2. Computing model of vehicle

Generally the one, two or three dimensional multi-body computing model of the vehicle can be adopted for the modelling of dynamic effect of moving vehicles on the road structures. The plane computing model of a lorry was introduced in [1]. For the purpose of this paper the space computing model of the lorry Tatra was adopted, Fig. 1.

Figure 1: Space computing model of the lorry Tatra

The computing model of vehicle has 15 degrees of freedom, 9 mass and 6 mass less. The mass less degrees of freedom correspond to the vertical movements of the contact points of the model with the surface of the runway. The vibration of mass objects of the model is described by the 9 functions of the time \( r_i(t) \), (\( i = 1-9 \)). The mass less degrees of freedom are coupled by tire forces \( F_j(t) \), (\( j = 5-10 \)) acting at the contact points.

3. Random road profile

The real road profile is of random character and it represents the dominant source of kinematical excitation of vehicle. A random road profile \( u(x) \) can be approximated from the known Power Spectral Density (PSD) in the form of

\[
S(\Omega) = S(1) \Omega^k,
\]

where \( \Omega = 2\pi/L \) in [rad/m] denotes the wave number and \( S(1) = S(\Omega_0) \) in [m²/(rad/m)] describes the value of the PSD at reference wave number \( \Omega_0 = 1 \) rad/m. According to the international standard ISO 8608, [5], typical road profiles can be grouped into classes from A to E. By setting the waviness to \( k = 2 \), each class is simply defined by its reference value \( S(1) \). Class A with \( S(1) = 1 \times 10^{-6} \) m²/(rad/m), class E \( S(1) = 256 \times 10^{-6} \) m²/(rad/m). Then the random road profile of a very good quality (class A, \( S(1) = 2 \times 10^{-6} \) m²/(rad/m)) was numerically generated by the equation

\[
u(x) = \sum_{\Omega} \frac{1}{2} S(\Omega) \Delta \Omega \cos(\Omega x + \phi) \cdot \]

In the equation (2) the \( \phi \) is the uniformly distributed phase angle in the range between 0 and \( 2\pi \). The results of a solution can be presented in time or frequency domains. The road profile was created separately for the left and for the right track. 10 240 samples with length step 0.01 m were generated in every profile. The demonstration of the road profile in the left and right track is shown in Fig. 2.

---

*This work was supported by Grant National Agency VEGA, project No. G1/0259/12.
4. Solution in time domain

The equations of motion of vehicle computing model are derived in the form of ordinary differential equations. The unknown functions \( r_i(t) \), \( i = 1-9 \) are solved numerically by the use of Runge-Kutta 4th order step-by-step integration method in the environment of program system MATLAB. Then the tire forces \( F_{j}(t) \), \( j = 5-10 \) are calculated. The time history of the tire force \( F_{5}(t) \) under right front wheel is shown in Fig. 3. The time history of the tire force \( F_{8}(t) \) under right rear wheel is shown in Fig. 4.

5. Solution in frequency domain

Passing from time to frequency domain can by carried out by two ways:

1. Transforming of equations of motion to frequency domain by some integral transform (Laplace or Fourier) and solving the equation in frequency domain [2].

2. Transforming the result obtained in time domain to frequency domain by a Discrete Fourier Transform (DFT) and processing the results in frequency domain.

The Power Spectral Densities (PSD) and Frequency Response Functions (FRF) or Power Response Factors (PRF) are analysed in frequency domain. The PSD of road unevenness \( u_{5} \) under right front wheel is plotted in Fig. 5. The dominant peak corresponds to the frequency 0.122 Hz. The wavelength \( L = 81.96 \text{ m} \) is coupled with this frequency.

After transforming the time records of tire forces to frequency domain their PSD can be calculated. When the road unevenness \( u_{5} \) is assumed as the source of kinematical excitation of vehicle the FRF or PRF of tire forces can be defined as a ratio of PSD of tire force and PSD of road unevenness \( u_{5} \). The obtained PRF is jagged. The sequential smoothing of such functions is needed to obtain the theoretical PRF. The LabVIEW system and program DIAdem was used on smoothing. The non-smoothed, smoothed and theoretical courses of PRF of tire force \( F_{5} \) under right front wheel of vehicle is plotted in Fig. 6.

References


Ultrasonic modulated waves for diagnostics of concrete, experimental study

Błażej Meronk¹, Krzysztof Wilde²
¹,²Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: blazej.meronk@wilis.pg.gda.pl¹, krzysztof.wilde@wilis.pg.gda.pl²

Abstract

The paper presents the experimental study on the diagnostics of concrete elements with the use of nonlinear acoustic effects. The tests were conducted on a concrete plate subjected to ultrasonic waves modulated with and without an additional low frequency actuator. The experimental results showed that the new method based on the direct modulation of diagnostic ultrasonic waves also provides sub-harmonic frequencies that indicated the presence of damage. The new method does not require a low frequency actuator for generation of low frequency oscillations, and therefore, might be more suitable for practical application.

Keywords: nonlinear acoustics, concrete elements, damage detection, ultrasonic waves

1. Introduction

The paper presents an experimental evaluation of two nonlinear acoustic modulation damage detection techniques on reinforced concrete plates. The first method is based on measuring modulation of ultrasonic wave by low frequency oscillations. The method can be called nonlinear wave modulation spectroscopy (NWMS) [e.g., 1]. The second method is based on measuring harmonic distortion of primarily modulated ultrasonic wave and the method is called a primarily modulated nonlinear wave spectroscopy (PMNWS). The study are devoted to understand the effects that occurs in nonlinear elastic materials during propagation of modulated ultrasonic waves. The nonlinear effects can be a very good tool to detect the presence of cracks that cause local nonlinear behaviour of the material. The review of the nonlinear acoustic based methods has been published by Broda et al. [2].

2. Experimental setup

The experimental setup consists of the reinforced concrete plate loaded with controllable static force, shaker to induce dynamic forces and system for generation and recording the diagnostic ultrasonic waves (Figure 1). The tested concrete specimens had the same size of 1700x300x80 mm and were made of concrete B20/25 with reinforcement of diameter 6 mm made of steel AIII-N. The static load system is used to engender cracks and control their expansion. The static force is subjected to the concrete specimen with use of pneumatic closed loop system that provides possibility of very precise control of the force during long time interval. The low frequency oscillations were generated with electrodynamic shaker. The ultrasonic system consists of piezoelectric transducers: one actuator and four sensors (Figure 2). The wave acquisition system requires pre-amplifiers, data acquisition system and computer responsible for storing and managing very large amount of data.

The ultrasonic testing was performed during slowly increasing static load from 0 to 4 kN. For NWMS method the shaker was used to generate the concrete specimen oscillations with frequency form 50 to 80 Hz and 3 levels of power. For each case of certain static load and the selected frequency and power of the shaker a few ultrasonic tests were conducted. For the diagnostic propose a single frequency continuous wave of 58 kHz was used. The locations of ultrasonic recordings taken at different levels of cracks evolution are given in (Figure 3).

Figure 1: Specimen on the test bench

Figure 2: Experimental setup

Figure 3: Example of the force-displacement relation with position of the ultrasonic tests denoted by circles
The experiments for PMNWS method are using modulated ultrasonic waves that are generated directly by piezoelectric transducer without using the electrodynamic shaker. Single frequency continuous 58 kHz wave was replaced by amplitude modulated ultrasonic wave. The carrier wave frequency was 58 kHz and modulation frequency was 50 Hz. The experiments using continuous wave and modulated wave were performed simultaneously using the same concrete sample.

3. Selected results

3.1. Nonlinear wave modulation spectroscopy (NWMS)

The analysis of the recorded ultrasonic signals is conducted in frequency domain. The sideband energy at frequencies shifted by multiple oscillation frequency 50 Hz reveals amplitude modulation of ultrasonic wave. The comparison of the spectra of undamaged and damaged specimen is shown in (Figure 4) and (Figure 5). The amplitudes of the sidebands are higher when specimen is damaged.

Figure 4: NWMS spectrum, undamaged sample

Figure 5: NWMS spectrum, damaged sample

3.2. Primary modulated nonlinear wave spectroscopy (PMNWS)

The post processing and signal analysis for NWMS method is the same as for the NWMS method. The undamaged sample, on spectra plots, show characteristic three peaks arranged in symmetrical fashion (Figure 6). The presence of damage and resulting nonlinearity in a cracked region of the concrete element distorts this pattern (Figure 7). The sideband on the left side central frequency is higher than sidebands of the right side.

Figure 6: PMNWS spectrum, undamaged sample

Figure 7: PMNWS spectrum, damaged sample

4. Final remarks

The paper presents the experimental study on two ultrasonic diagnostic methods: NWMS and PMNWS. The first one uses the low frequency actuator to provide modulation of ultrasonic waves used for diagnostic purposes. The second method, proposed in this paper, uses modulated waves generated directly by the piezoelectric actuator. The experimental results showed that the new method also provides sub-harmonic frequencies useful for detection of damage. The new method does not require a low frequency actuator, and therefore, might be more suitable for practical application.

References


Numerical analysis and in situ tests of Grot Rowecki bridge in Warsaw

Mikołaj Miśkiewicz¹, Łukasz Pyrzowski², Krzysztof Wilde³, Jacek Chróścielewski⁴

¹,²,³,⁴ Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
E-mail: mmisk@pg.gda.pl ¹, lpyrzow@pg.gda.pl ², wild@pg.gda.pl ³, jchrost@pg.gda.pl ⁴

Abstract

The paper presents the FEM analysis of the rebuilt Grot-Rowecki bridge over Vistula river in Warsaw. The bridge was built in 1981 and since 2012 is under reconstruction. The bridge has seven spans and consists of two independent structures with the longest spans of 120 m. After reconstruction the bridge will be over 10 m wider. Numerical analysis has been conducted on a global and local FEM models. These models are defined so as they strictly comply with global structural bridge behaviour and satisfy fundamental rules of mechanics. A detailed analysis makes it possible an insight into the issues like the local accumulation and concentration of stresses, local stability, local dynamic factors. The overview of the in situ tests conducted on the bridge is presented.

Keywords: Grot-Rowecki bridge, local models, FEM, test loading

1. Introduction

The Grot-Rowecki bridge over Vistula River in Warsaw is one of the most important bridges in Warsaw. Before reconstruction it had 2×4 lanes and it allowed to cross Vistula river hundreds of thousands of cars every day (Fig. 1). The bridge consists of two independent steel 7-span structures with static scheme of continuous beam and total length of 646 m (Lₜ = 75 + 3×90 + 2×120 + 60 m). The longest spans with length of 120 m is designed as box superstructure. Other spans have steel girders. The bridge was built in 1981 and designed in accordance to the code PN-66/B-02015.

According to the modernization plans of entire S-8 express road, the bridge was not wide enough and had insufficient strength therefore it had to be rebuilt. The major structural change, in the new bridge, is addition of pre-stressed cables that allowed to widen each structure by truss girders with additional lane and footpath (Fig. 2).

2. FEM models

To verify the new bridge capacity and its agreement with design assumptions the FEM calculations were performed using two levels of the structure description – global and local [1]. The purpose of analysis at the global level was to check the overall behaviour of the entire structure and to determine an envelope of generalized internal forces. The local simulations provide a detailed information concerning stress distribution in the selected areas of the structure. The local approach is focused on the elements in question i.e. orthotropic deck plate with a detailed description of ribs and nodes of newly added truss members.

On the basis of the global description of the bridge three zones of orthotropic were selected and analysed locally. In these models shell finite elements were used to describe bridge plate, top plate ribs in the impact zone of load, webs of girders and traverses and the bottom plate. Other structural elements were described as beam elements. The most complex detailed model of the bridge section consists of about 1.5 million degrees of freedom (Fig. 3).

Figure 1: Grot-Rowecki bridge over Vistula river in Warsaw (www.wikipedia.org)

According to the modernization plans of entire S-8 express road, the bridge was not wide enough and had insufficient strength therefore it had to be rebuilt. The major structural change, in the new bridge, is addition of pre-stressed cables that allowed to widen each structure by truss girders with additional lane and footpath (Fig. 2).

Figure 2: Redesigned cross section of the bridge

On the basis of the global description of the bridge three zones of orthotropic were selected and analysed locally. In these models shell finite elements were used to describe bridge plate, top plate ribs in the impact zone of load, webs of girders and traverses and the bottom plate. Other structural elements were described as beam elements. The most complex detailed model of the bridge section consists of about 1.5 million degrees of freedom (Fig. 3).

Figure 3: Visualization of one of the local FEM model

In order to validate the adopted local formulations with global approach for each of the detailed models a number of numerical tests were carried out. They aimed at verification of compliance of resultant cross-sectional internal forces as well as the verification of the stress distribution in the regions described using the same FE types in global and local approach [2].
3. Selected results for bridge deck

The detailed deck analysis is required to check the stress levels and their agreement with design assumptions. There have been concerns that the loads caused by the wheels of the code vehicle may lead to dangerous, local increase of stresses in deck plate and in ribs below it. The extreme values of stress for the two variants of FEM models are presented in Table 1.

Table 1: Numerical results

<table>
<thead>
<tr>
<th>Part of the deck</th>
<th>Global model [MPa]</th>
<th>Local model [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plate</td>
<td>186</td>
<td>239</td>
</tr>
<tr>
<td>Rib</td>
<td>390</td>
<td>244</td>
</tr>
<tr>
<td>Traverse</td>
<td>115</td>
<td>161</td>
</tr>
</tbody>
</table>

The obtained results indicate redistribution of stresses in the bridge deck plate. The significant decrease of peak stress in the ribs with respect to the results obtained in a global approach is observed. Finally it is noted that in some critical configuration of loads the local stresses in steel may be exceeded by 6.7% for the plate and 8.9% for the ribs. On the other hand, it should be noted that the stress concentrations are local effects due to the code vehicle and they will never occur in the reality. Finally it was decided that the deck load capacity is sufficient and the bridge is designed properly. It was suggested that the safety level of the bridge might be increased by installation of structural health monitoring system (SHM) [3].

4. In situ load test

According to the Polish bridge design code road bridges with spans over 20 m and all railway bridges, before they are given to public use, require the in situ load tests. The goal of the experimental work is to check the agreement of the true structural bridge behavior (static and dynamic) with the design assumptions. The analysis is based on comparison of the measurement data with FEM results [4]. The numerical model should faithfully reflect all important structural elements, material selections, the structural member joints, the proposed construction phases and the structure sensitivity to external factors. In case of Grot-Rowecki Bridge both global and local models are used to verify its theoretical behavior on the experimental data.

Figure 4 shows one of the in situ load case conducted with use of four axle trucks of weight 332 kN each. First, the numerical analysis of bridge behavior under the load of 30 trucks has been conducted. In the next step field tests were carried out and the real behavior of the structure was measured. The strains and displacements of the bridge under static and dynamic loading were recorded. The detailed measurements of the stress distribution in the nodal truss plates were also conducted (Fig. 5). During the dynamic tests accelerations at representative points on the bridge deck were recorded.

The measured values of displacements, stresses and accelerations are in a good agreement with the corresponding numerical values. The relative errors between the measured and numerical results are about 7%.

5. Summary

In bridge design practice, in order to perform the numerical analysis of the structure, the beam grid scheme is commonly used. However, the results from the theoretical analysis and the in situ measurements of Grot-Rowecki Bridge in Warsaw highlight the need of applying more advanced numerical models for getting a better understanding of the real behaviour of the structure. This statement is particularly important for the analysis of local structural elements of a large bridge structure. The local analysis with detailed FEM models must take into account need to satisfy the fundamental rules of the mechanics in relation between the local and global model. In case of Grot-Rowecki Bridge, a good agreement between the numerical results, for both the global and local approach, and the data recorded during the in situ tests were obtained.

References

Experimental tests and numerical simulations of a full scale composite sandwich segment

Mikołaj Miśkiewicz1, Karol Daszkiewicz2, Tomasz Ferenc3, Wojciech Witkowski4, Jacek Chróścielewski5

1,2,3,4,5 Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland

*e-mail: mmisk@pg.gda.pl, kardaszk@pg.gda.pl, tomferen@pg.gda.pl, wojwit@pg.gda.pl, jchrost@pg.gda.pl

Abstract

In the paper experimental tests and numerical simulations of full cross-section segment of ultimately designed foot- and cycle-bridge are presented. Experimental tests were conducted on an element with length reduced to 3 m and unchanged (target) cross-sectional dimensions. The external skin of a structure is a GFRP laminate while an internal core is PET foam. Several quasi-static tests were performed using hydraulic cylinder to generate vertical loads, and using a specially designed system of rods to generate horizontal loads applied to the structure transversally and longitudinally. The results of numerical simulations conducted separately in two FEM environments are shown. They serve as a basis for the design process of target foot- and cycle-bridges.

Keywords: experimental tests, composite foot-and-cycle bridge, GFRP laminate, PET foam

1. Introduction

The aim of the project FOBridge (PBS/B2/6/2013), conducted in consortium consisting of Gdańsk University of Technology, Military University of Technology in Warsaw and a private company ROMA Sp. z o. o. [1], is to design and manufacture a single span foot- and cycle-bridge made entirely of composite materials and manufactured as one element in the infusion process.

In order to design innovative structures some experiments are required. They can be divided into two groups: experiments determining material parameters and experiments to validate correctness of the assumptions made during numerical simulations. In the paper validation tests of a numerical model of a full cross-section segment are presented.

2. Validation segment

The validation segment was constructed as a sandwich structure with an external skin of GFRP laminate and an internal core of PET foam. The cross-sections for the segment and the target foot- and cycle-bridge are the same. The total length of the segment is 3 m, while the theoretical span length was established as 2.5 m. The usable width is 2.59 m and the height of handrails is 1.3 m. The segment is shown in Fig. 1. The segment was placed on rubber bearings of dimensions 30x30x3 cm. The support areas were additionally strengthened.

3. Conducted experiments

Several quasi-static tests were performed for the composite sandwich segment. The load schemes were classified into four categories: A – vertical load applied by a hydraulic cylinder on the platform or the top of walls, B – horizontal load applied on the top of walls transversally simulating walls bending, C – horizontal load applied on the top of walls longitudinally simulating walls compression, D – combination of the above loads, the example is shown in Fig. 2. Additionally, the scheme S with a crowd loading was carried out.

Figure 2: The example combined load designated as D3b

The load schemes B, C and D were conducted using a specially designed system of fixing elements and rods. On the basis of preliminary FEM numerical simulations the load value for a linear range was estimated. In the calculations the failure index according to Tsai-Wu hypothesis [2] for skin and compression principal stress for foam was controlled.

Several measurement points were regarded during the experiments: 60 strain gauges, 18 displacements sensors and 4 acoustic sensors. The load value was applied to a segment in many steps (see Fig. 3), due to creep of a sandwich structure. During validation tests of the numerical model, the values in strain gauges were monitored and compared with preliminary numerical results.

Figure 1: The validation segment – a cross-sectional view

* The study is supported by the National Centre for Research and Development, Poland, grant no. PBS1/B2/6/2013.
4. Numerical simulations

Numerical simulations were performed separately by means of two FEM programs. The models were built using 4-node single and multilayered shell elements and 8-node solid elements. Material parameters determined in the previous experiments [3] or values given by producers were assumed in the numerical analysis. The mesh was generated with a total number of about 196,000 elements, and a total number of about 153,000 nodes.

A detailed analysis of a the load case A4 (Fig. 4) is presented. In order to obtain more precise results for the numerical model, the support zones (i.e. material parameters of the bearings) were modified (updated) after validation tests. For a maximum load value 50 kN numerical and experimental values of strains are compared in Table 1. The placement of selected strain gauges is presented in Fig. 4. In addition, Fig. 5 and Fig. 6 show the comparison of two experimental curves with the numerical curve for strain gauge T5/4 and inductive sensor U9/3, respectively.

Table 1: Comparison of numerical and experimental strains

<table>
<thead>
<tr>
<th>Strain gauge</th>
<th>Experimental [μm/m]</th>
<th>Numerical [μm/m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>T9/1</td>
<td>668</td>
<td>635</td>
</tr>
<tr>
<td>T4/3</td>
<td>137</td>
<td>160</td>
</tr>
<tr>
<td>T6/3</td>
<td>1240</td>
<td>1589</td>
</tr>
<tr>
<td>T8/3</td>
<td>167</td>
<td>184</td>
</tr>
<tr>
<td>T15/3</td>
<td>835</td>
<td>760</td>
</tr>
<tr>
<td>T9/5</td>
<td>585</td>
<td>631</td>
</tr>
</tbody>
</table>

5. Summary

In the paper some experimental tests and numerical simulations of a full scale composite sandwich segment have been presented. Validation of a numerical model is a difficult task, due to complexity of a structure and great importance of the support and load zones in load-carrying process. The experimental values of strains and displacements are in good agreement with the numerical results. A small discrepancy between the results may be caused by a nonlinear behaviour of the bearings.

In April 2015 a full-scale bridge with span length of 14m was erected at Gdańsk University of Technology. In the following 6 months it is being thoroughly tested and monitored by a SHM system [4].

References


Numerical analysis of the carpentry joints applied in traditional wooden structures

Anna Mleczek¹, Paweł Kłosowski²

¹,² Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
email: ammlecz@pg.gda.pl¹, klosow@pg.gda.pl²

Abstract

The paper concerns the numerical analysis of carpentry joints made of spruce wood. The material parameters of wood have been calculated on the basis of nine independent material constants (an orthotropic material). The contact zone between the individual elements of the connection has been determined. The aim of the paper is selection of the areas of greatest stress in the planes of contact, and thus determination of vulnerable damage zones.

Keywords: carpentry joints, wood, finite element models – solid elements, contact

1. Introduction

Nowadays wooden structures are not so common as steel or concrete structures. This is essentially, due to the properties of material and the price of wood construction. Nevertheless, there are many wooden buildings, mostly historical, requiring maintenance, renovation and reinforcement of the existing elements. This paper is focused on carpentry joints used in traditional, mainly historical wooden structures. Unfortunately, the lack of literature in the field of mechanical behaviour of such connections is still observed. The engineering studies of carpentry joints are carried out primarily during historical analyses and roof constructions using photoelasticity [1, 2, 3]. These analyses are related to the connections with the use of a wooden stem. These studies are different examples of a solution compared to traditional carpentry joints, the subject of the paper.

The topic of carpentry joints is also extremely important in view of a growing interest of architects and engineers in structures of wood or wood components.

2. Geometry of the carpentry joint

The geometry of the analysed exemplary carpentry joint consisting of three timbers is shown in Fig. 1.

Due to the nature of the work, the connection has been built using three-dimensional solid elements. The following boundary conditions has been adopted:
- at the end of a single timber - blocked translations in three directions (X, Y, Z),
- at the end of two timbers - blocked translations in two directions (X and Z) and the applied surface load on the Y direction.

The contact zones have been defined and presented in Fig. 2.

3. Material parameters of wood

The anatomical directions in a wood are distinguished depending on the grain of the annual increment [6]. In the literature concerning the properties of the wood is applied literal notation:
- $R$ – radial direction,
- $T$ – tangential direction,
- $L$ – longitudinal direction to the surface of each layer of the fibres.

In turn, in the theory of elasticity is used index notation:
- $x_1$ – radial direction,
- $x_2$ – tangential direction,
- $x_3$ – longitudinal direction.

The anatomical directions of wood are presented in Fig. 3.
Wood is an orthotropic material, described by twelve material constants. Due to symmetry with respect to the main diagonal of the constitutive matrix, three reciprocal relations must be satisfied:

\[ \frac{v_{ij}}{E_i} = \frac{v_{ji}}{E_j}, \quad i, j = 1, 2, 3. \]  

(1)

On this basis, nine independent material constants are distinguished [4, 6]. The analysed carpentry joint is made of spruce, which material parameters are presented in Table 1.

<table>
<thead>
<tr>
<th>Material constants</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_{x}$</td>
<td>849</td>
<td>[MPa]</td>
</tr>
<tr>
<td>$E_{y}$</td>
<td>468</td>
<td>[MPa]</td>
</tr>
<tr>
<td>$E_{z}$</td>
<td>10890</td>
<td>[MPa]</td>
</tr>
<tr>
<td>$G_{12}$</td>
<td>33</td>
<td>[MPa]</td>
</tr>
<tr>
<td>$G_{23}$</td>
<td>664</td>
<td>[MPa]</td>
</tr>
<tr>
<td>$G_{31}$</td>
<td>697</td>
<td>[MPa]</td>
</tr>
<tr>
<td>$v_{12}$</td>
<td>0.372</td>
<td>[-]</td>
</tr>
<tr>
<td>$v_{23}$</td>
<td>0.435</td>
<td>[-]</td>
</tr>
<tr>
<td>$v_{31}$</td>
<td>0.025</td>
<td>[-]</td>
</tr>
</tbody>
</table>

In order to define the orthotropic material in the MSC Marc all the components of the constitutive matrix have been introduced individually based on the data presented in Table 1.

4. Results

The geometrically nonlinear static analysis of the carpentry joint has been carried out taking into account the contact phenomenon between the individual elements for the material data presented in the previous section. Fig. 4 presents the obtained deformation of the system. In turn, Fig. 5 presents reduced stress distribution in the connection corner according by Huber - von Mises – Hencky criterion.

References


Real-time hybrid simulation using materials testing machine and Finite Element Method

Waldemar Mucha*
Institute of Computational Mechanics and Engineering, Faculty of Mechanical Engineering, Silesian University of Technology
Konarskiego 18A, 44-100 Gliwice, Poland
e-mail: waldemar.mucha@polsl.pl

Abstract

The paper presents an algorithm for a real-time hybrid testing using Finite Element Method. A hybrid testing, also commonly referred as hardware-in-the-loop simulation, is a method that allows to investigate dynamic properties of some complex mechanical structures or systems. The testing involves creating two models: the experimental model (a physical component of the tested structure or system) and the virtual model (a numerical model of the rest of the tested structure or system). The paper describes an algorithm for hybrid testing where the experimental model is a physical component of a mechanical structure mounted in a dynamic materials testing machine and the machine is controlled by an advanced microcontroller with the virtual model implemented in its algorithm. The virtual substructure of the tested object is modelled using Finite Element Method. An example is given where the tested mechanical structure is a truss and one of its elements is mounted in the testing machine.

Keywords: hybrid testing, hardware-in-the-loop-simulation, Finite Element Method, real time

1. Introduction

Hybrid testing (hardware-in-the-loop simulation) is a method that allows to investigate dynamic properties of mechanical structures or systems. The method consists of creating two models strictly related to each other: the experimental model which is a physical component of the tested structure and the virtual model which is a numerical representation of the rest of the tested structure. The popularity of hybrid testing technique has grown rapidly since its conception (in the turn of 1960s and 1970s) because of its undeniable advantages like the ability to test complex structures containing elements that are difficult or even impossible to model numerically, the possibility to experimentally test only a component of a structure which sizes make impossible to test it as a whole, or significantly lower costs compared to shaking table tests [1, 3].

While building an experimental model, the physical component of the structure is usually constrained and attached to one or more dynamic actuators that load the physical components [3]. The principal idea is to mount the physical component of the tested structure to a dynamic materials testing machine. This solution comes with some important advantages as such machines often have a wide force range, are able to perform high-speed tests and allow easy measurements of forces and displacements with embedded measurement systems. The testing machine can be controlled with an advanced microcontroller with the virtual model of the hardware-in-the-loop (HIL) simulation implemented in its algorithm. The virtual subsystem of the tested object can be modelled using the Finite Element Method (FEM). The tasks of the microcontroller is to generate signals controlling the testing machine, perform computations on the virtual model and acquire data from built-in measurement systems of the testing machine.

2. The algorithm

The algorithm of the hybrid simulation using FEM is based on a modified FEM matrix equation of motion:

\[ \mathbf{M} \ddot{\mathbf{u}} + \mathbf{C} \dot{\mathbf{u}} + \mathbf{K} \mathbf{u} + \mathbf{r}^E = \mathbf{f} \] (1)

in which \( \mathbf{M} \), \( \mathbf{C} \) and \( \mathbf{K} \) are mass, damping and stiffness matrix of the virtual model, respectively, \( \mathbf{r}^E \) is the vector of forces developed in the experimental model (depending on structural displacement, velocity and acceleration), \( \mathbf{u} \) is the displacement vector and \( \mathbf{f} \) is the force vector [2,3,4].

![General hybrid testing algorithm](image)

* Acknowledgement: The research is partially financed from project BKM-504/RMT-4/2014
Figure 1 presents a general algorithm for hybrid testing of structures. The algorithm requires iterative approach. The equation (1) has to be solved for each time step \( i \). In order to solve the equation (1) explicit or implicit methods can be used. Explicit methods (e.g. Central Difference Method) are usually computationally less demanding but require small time steps \( \Delta t \) as they are conditionally stable. On the other hand implicit methods (e.g. Newmark Method) are often unconditionally stable (which means that larger time steps can be used) but require more calculations to be performed [3, 4].

The algorithm presented in Fig. 1 presents only the idea of the hybrid simulation and does not detail additional operations that have to be performed, e.g. compensating the delay between sending the displacement command to the testing machine and the response of the machine. It is also possible that the time step used to solve the equation (1) in the virtual model and the time step of successive operations of the testing machine in the experimental model are different. In such case some extrapolation or interpolation methods can be used [3].

3. Numerical example

A truss shown in Fig. 2 is considered as the numerical example. Young modulus is equal to 200 GPa, density 7800 kg/m³, cross section of all the members is circular with diameter 10 mm. Member 11 of the truss represents the experimental model and other members are the virtual model.

\[
C = 5M
\]  

(2)

The experimental model is simulated numerically by an external function that returns residual force of member 11 depending on its elongation. It is assumed that member 11 is non-linear elastic and the relationship between the force and elongation is approximated by the following equation:

\[
N = 8.893\Delta l^4 - 20.044\Delta l^3 - 2.225\Delta l^2 + 35.25\Delta l - 0.209
\]  

(3)

In equation (3) \( N \) denotes the force in kN and \( \Delta l \) the elongation in mm. Inertial and damping forces related to horizontal degrees of freedom of member 11 had to be included in the virtual model.

The FEM equation of motion (1) was solved using Central Difference Method. Because the highest natural frequency of the truss is 2377.6 Hz, the time step must not exceed 0.134 ms. In the simulation time step was equal 0.1 ms.

Figure 3 presents the excitation force \( F(t) \) as the function of time.

4. Conclusion

Implementing FEM and using materials testing machines in HIL simulations may lead to important advantages. FEM has a wide area of applications and testing machines are often easier to setup as the experimental model than using classical actuator approach.

The presented example proves the correctness of the algorithm shown in Fig. 1 as the results were verified with classical dynamic FEM algorithm without substructuring.

The work on experiment with materials testing machine is ongoing and results are planned to be presented in the future.

References

Experimental evaluation of wavelet based damage monitoring of a reinforced concrete frame

Marek Nalepka, Zbigniew Zembaty, Seweryn Kokot

Faculty of Civil Engineering, Opole University of Technology
Katowicka 48 45-061 Opole, Poland

e-mail: m.nalepka@po.opole.pl, z.zembaty@po.opole.pl, s.kokot@po.opole.pl

Abstract

Possibilities for damage diagnosis by means of continuous wavelet transforms are investigated on an example of seismic response of a reinforced concrete (r/c) frame subjected to shaking table tests. The linear and nonlinear responses are compared using various wavelets. The results demonstrate characteristic development of wavelet ridge and peak patterns following accumulation of damage in the r/c frame which is a good prognostic for wavelet application in on-line monitoring of r/c structures.

Keywords: vibrations, Structural Health Monitoring, reinforced concrete, continuous wavelet transform

1. Introduction

Wavelet transform plays a significant role in vibration-based structural damage detection (see e.g. [3]). Respective damage detection can for example be obtained through a spatially applied wavelet [1]. However the wavelet transform can also be applied in time domain. In this case the specific differences of the linear structural response compared to nonlinear response with energy dissipation in damaged state are investigated with the application of wavelet transform. Such an approach was already investigated [2]. Its experimental validation is a critical issue now.

The lecture proposed for the 3rd PCM-CMM Congress will present early results of the application of Morlet wavelet to evaluate data acquired during a shaking table experiment devoted to modal analysis of an R/C frame in various damage states [4].

2. Description of the experiment

The analysed reinforced concrete frame was one of two frames taken in an experiment described in detail in a paper by Zembaty et. al [4]. The frame was subjected to artificially generated, two-component seismic signals on a shaking table (see Fig. 1). The excitation intensity was changed during the tests, stage by stage using a multiplier of the original signal. This resulted in linear excitations at the beginning of the tests until strong damaging effects. Each phase of strong ground motion was interlaced with a series of diagnostic small excitations (sweep sine, random tests, time history excitations) aiming at retrieving evolution of the modal parameters of the damaged frame described in detail in Ref. [4]. From this experiment two acceleration records of sensor A22 (Fig. 1) were chosen for the wavelet analysis. From the early stage, linear response of intact frame with maximum value of 35,6 cm/s² for which the dynamic parameters of the frame equalled: T₁=0,21 s (translational mode along x axis), T₂=0,17 s (along y axis) and T₃=0,10 s (torsion). The second record represents nonlinear dynamic response with 319,0 cm/s². Respective, “linearized” natural frequencies of the frame (taken from post-damage diagnostic tests) equalled: T₁=0,35 s, T₂=0,28 s and T₃=0,13 s.

![Figure 1: Analysed frame with location of accelerometer A22 and seismic records of accelerations (along x and y axis)](image-url)
3. Application of Continuous Wavelet Transform to observe damage evolution

The continuous wavelet transform (CWT) of a function, represents the function or the time history $f(t)$ as a sum of dilated (by a scale parameter $a$), the scale in this case related to the frequency of wavelet, and time-shifted (by a shift parameter $b$) wavelets. Because wavelets are localized waves that span a finite time duration, CWT can represent time-varying characteristic of $f(t)$. The CWT is defined as follows:

$$Wf(a,b) = \int_{-\infty}^{\infty} f(t) \frac{1}{\sqrt{a}} \psi\left(\frac{t-b}{a}\right) dt$$  \hspace{1cm} (1)

where $\psi(t)$ is the so-called mother wavelet chosen depending on the wavelet application. The mother wavelet is extended by scale parameters and translate over the $f(t)$ by shift parameter $b$, which creates basic functions called daughter wavelets.

In Figures 2 and 3 selected results of the application a Morlet wavelet as the mother wavelet are presented. The specific puls like shape and a good time-frequency resolution of the Morlet wavelet makes its application in this case particularly plausible. Scale of daughter wavelet of Morlet wavelet correspond pseudofrequency to the natural frequency of structure. Comparing the wavelet map of the linear response (Fig. 2) with the non-linear one (Fig. 3) we notice that as the damage extent increases, the peaks of ridges of wavelet coefficients shifts both in time and scale. Changes in scale represent loses of high-frequency components due to loss of stiffness. As the intensity of damage increases the wavelet coefficients spread in time.

4. Conclusion

Wavelet analysis of all the response records of the r/c analyzed frame reveals clear, characteristic changes in the development of the wavelet ridges and peak patterns from the moment the structure starts to accumulate damage. This validates the qualitative positive assumption of applying the wavelet approach in the damage monitoring of r/c structures under strong, damaging dynamic excitations. The aim of the ongoing research is to build and test software capable of raising alarms of the imminent structural damage. At this stage various methods are tested. Detailed results of the quantitative comparisons will be reported during the Congress.

References


Comparative analysis of compound annular plates vibration on the basis of numerical and experimental studies

Piotr Nazarko1, Leonard Ziemiański2, Stanislaw Noga2, Tadeusz Markowski3

1,2 Faculty of Civil and Environmental Engineering and Architecture, Rzeszow University of Technology
Powstancow Warszawy 12, 35-959 Rzeszow, Poland
e-mail: pnazarko@prz.edu.pl1, ziele@prz.edu.pl2

3,4 Faculty of Mechanical Engineering and Aeronautics, Rzeszow University of Technology
Powstancow Warszawy 12, 35-959 Rzeszow, Poland
e-mail: nog@prz.edu.pl3, tmarkow@prz.edu.pl4

Abstract

The article discusses the impact of geometrical discontinuities on the transverse vibration of compound annular plates. For this purpose, a numerical analysis was performed on both a solid model and models with holes of different diameters. However, since a number of assumptions has been adopted in the FEM model, the results obtained were then verified by experimental studies. In this investigations, non-contact measurement were used and different types of extortion were studied. The comparison has confirmed that the results of numerical analysis remains at a fairly good level of compliance.

Keywords: transversal vibration, annular systems, numerical solutions, experimental investigations

1. Introduction

The problem of transverse vibration of compound annular and circular plates are studied by many researchers and centres of research and development [1–6]. This is due to the reason that some rotating systems, e.g. toothed gears or railway wheels can be considered as a circular or annular plates [1,5], where their shape and dimensions depend on the design of these systems. Experimental studies of the plate dynamics were initiated by Chladni [6], who developed empirical method of finding nodal lines on free circular plates. The fundamental theory of vibration of circular symmetric plates is elaborated in a number of monographs by, for example Leissa [4], and others. Transverse vibration of circular plate with disturbed geometry (by holes) are considered in the papers [2,3]. Studies of the toothed gears transverse vibration as an annular rotating plate are discussed in works [1,5].

In the work the discussion are focused on the experimental verification of simplified FE models of toothed gears which are treated as annular plates with geometrical discontinuities. These FE models are effectively used in papers [1,5] to identify the deformed normal modes of gears.

2. Formulation of the problem

The set of analysed objects consisted of four annular plates. Overall dimensions of considered systems are related with the geometry of the arbitrary chosen toothed gear (described in [1]) which are shown in Fig. 1. The first object is a solid plate with the rim. In other models of plates there are holes of various diameters do (see Fig.1) which are given in the header of Table 1. The problem of the free transverse vibration of systems under consideration is solved by the finite element method. As it is known, the nodal lines associated with nodal circles and nodal diameters are deformed due to through holes. The primary purpose is to confirm this fact in experimental studies.

3. Numerical analysis

In the elaboration of the FE models of considered systems the cyclic symmetry feature is used. As shown in Fig. 1, each model consists of five homogeneous sectors. Each of these segments is meshed using standard procedures of the ANSYS program. A 3–D solid mesh is prepared and the ten node tetrahedral element (solid187) with three degrees of freedom in each node is used to realise each sector. During the mesh generation process, it is assumed that the maximum length of each element’s side needs to be no more than 2 [mm]. Parameters characterising the systems used in the calculations are as follow: Poisson’s ratio ν = 0.28, Young’s modulus E = 2.1·1011 [Pa] and mass density ρ = 7.85·103 [kg/m3]. In accordance with the plate vibration theory [3, 4, 6], the particular natural frequencies of vibration are denoted as $\omega_m$ where $m$ refers to the number of nodal circles and $n$ is the number of nodal diameters. The differences between the experimental data and the FE solutions will be applied as they are applied in [5]

$$
\varepsilon = \frac{\omega^e - \omega^p}{\omega^p} \times 100 \% \tag{1}
$$

where $\omega^e$ is the natural frequency from the FE model, while $\omega^p$ is the natural frequency from the experimental investigation.

The representative results of the numerical calculations related to the natural frequencies of the FE models under
consideration are shown in Table 1. An exemplary mode shape related to the frequency $\omega_{25}$ is shown in Fig. 2a.

Table 1: Values of the natural frequencies and the frequency error

<table>
<thead>
<tr>
<th>$d_0$ [mm]</th>
<th>0</th>
<th>7.28</th>
<th>15.68</th>
<th>22.4</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Experimental investigations</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\omega_{22}$ [Hz]</td>
<td>7638</td>
<td>7254</td>
<td>7074</td>
<td>7278</td>
</tr>
<tr>
<td>$\omega_{24}$ [Hz]</td>
<td>7705</td>
<td>7633</td>
<td>7581</td>
<td>7642</td>
</tr>
<tr>
<td>$\omega_{21}$ [Hz]</td>
<td>14133</td>
<td>13413</td>
<td>14953</td>
<td>15059</td>
</tr>
<tr>
<td>$\omega_{26}$ [Hz]</td>
<td>17533</td>
<td>17328</td>
<td>17184</td>
<td>17480</td>
</tr>
<tr>
<td>$\omega_{25}$ [Hz]</td>
<td>19984</td>
<td>18871</td>
<td>19246</td>
<td>17707</td>
</tr>
<tr>
<td><strong>The FE solutions</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\omega_{22}$ [Hz]</td>
<td>7779</td>
<td>7708</td>
<td>7759</td>
<td>7753</td>
</tr>
<tr>
<td>$\omega_{24}$ [Hz]</td>
<td>7760</td>
<td>7746</td>
<td>7731</td>
<td>7806</td>
</tr>
<tr>
<td>$\omega_{21}$ [Hz]</td>
<td>15218</td>
<td>15046</td>
<td>14116</td>
<td>13887</td>
</tr>
<tr>
<td>$\omega_{26}$ [Hz]</td>
<td>17575</td>
<td>17522</td>
<td>17534</td>
<td>17903</td>
</tr>
<tr>
<td>$\omega_{25}$ [Hz]</td>
<td>19757</td>
<td>19373</td>
<td>19713</td>
<td>21481</td>
</tr>
</tbody>
</table>

| Frequency errors | $\varepsilon_{22}$ [%] | 1.9 | 6.3 | 9.7 | 6.5 |
| $\varepsilon_{24}$ [%] | 0.7 | 1.5 | 2.0 | 2.1 |
| $\varepsilon_{21}$ [%] | 7.7 | 12.1 | -5.6 | -7.8 |
| $\varepsilon_{26}$ [%] | 0.2 | 1.1 | 2.0 | 2.4 |
| $\varepsilon_{25}$ [%] | -1.1 | 2.4 | 6.5 | 3.6 |

Figure 2: Mode shape related to frequency $\omega_{25}$: (a) the FE solution, (b) experimental investigation

4. **Experimental investigations**

Dynamic tests of the investigated compound annular plates models, due to their relatively small size were the only possible through the use of non-contact measurements. Therefore a laser scanning vibrometer (PSV-400) was used to perform laboratory experiments. Through the hole in the center of models, each plate was mounted horizontally to the measuring position using double-sided spacer washers and a set screw. The laser head was placed directly above the studied models at a distance of 95 cm (Fig. 3). Additionally, in order to carry out automatic measurement, the outer rings of all tested models are equipped with piezoelectric transducers, which served as shakers.

The test results were also analysed in terms of the impact caused by the different sources of vibration excitation (modal hammer, electrodynamic exciter, piezoelectric transducers) and a variety of forms (pseudo-random noise, sweep).

5. **Conclusions**

Taking into account achieved results, it is clear that numerical studies can be effectively verified by experimental investigations. The results of these preliminary studies have also shown that, with respect to the solid plates, the introducing of holes decreased in principle the natural frequencies for the two lowest diameters, whereas in the case of the largest diameter, there may be notice some disturbances, causing even the increase in the frequency shift. However, this requires further analysis and more accurate comparisons of vibration forms, e.g. calculating the MAC factors.

**References**


Verification of the building FEM model on the basis of natural vibration measurements

Krzysztof Nepelski1, Ewa Blazik-Borowa2, Tomasz Łipecki3, Jarosław Bęc4
1,2,3,4 Faculty of Civil Engineering and Architecture, Lublin University of Technology
Nadbystrzycka 40, 20-618 Lublin, Poland
e-mail: k.nepelski@pollub.pl, e.blazik@pollub.pl, t.lipecki@pollub.pl, j.bec@pollub.pl

Abstract

The paper describes FEM modelling of a four-storey building of a mixed reinforced concrete and masonry structure. The model takes into account the actual work of a connection between reinforced concrete and brick elements by releasing the respective degrees of freedom in these connections. Modal analysis is carried out for the model of the building. Verification of the FEM model is based on the in-situ measurements of the vibration frequencies. Vertical vibrations of floors were excited during the in-situ tests. Accelerations of vertical vibrations were measured in the locations selected on the basis of technical documentation. Power spectral density functions and dominant frequencies of natural vibrations were estimated on the basis of the recorded time-dependent accelerations and used to verify the numerical model of the building.

Keywords: four-storey building, FEM model, frequency of natural vibrations, in-situ measurements, modal analysis

1. Introduction

FEM modelling of a building structure was described in [1,2]. Natural vibrations estimated in the real scale are often used as a verification of the FEM model mainly of high-rise buildings subjected to wind action or seismic loads [3-5]. The paper also presents a process of the FEM modelling of the four-storey apartment building. The work of the building was modelled in the linearly-elastic state, as in [2]. The FEM model was verified on the basis of natural frequencies identified on the basis of the in-situ tests.

2. Technical description of the four-storey building

The object consists of three buildings with dilatations between them along the whole height. The subject of numerical analysis is one of the segments of the building, considered for the FEM model (Fig. 1). The structure of the building is mixed of reinforced concrete and masonry. The walls in the underground section are made of reinforced concrete, the overground parts are made of brick and concrete. The thickness of walls is 25 cm. Additional reinforced concrete columns are applied in some walls. Ceilings are monolithic concrete with the thickness of 20, 22 and 30 cm. The object is supported on the lower edges of the basement walls, additionally on several columns.

Figure 1: FEM model of the building (different shades are related to different materials)

3. FEM model

The numerical model created in Abaqus 6.12, consists of plate and beam elements. Four-node plate elements of the S4R type were used to model walls, ceilings and binding joists. Columns in the basement were created as beam elements. Elastic material properties of the reinforced concrete elements were assumed as follows: \( \rho = 25 \text{ kN/m}^3 \), \( \nu = 0.2 \) [7]. Elastic modulus \( E = 34.9 \text{ GPa} \) was estimated from eq. (1).

\[
E = \frac{E_s J_s + E_c J_c}{J}
\]  

The following parameters were assumed for brick elements: \( \rho = 17 \text{ kN/m}^3 \), \( \nu = 0.25 \) and \( E = 4.5 \text{ GPa} \) [8].

The work of connections between reinforced concrete and brick materials in the FEM model should be adequately modelled. Joint connections were used between reinforced concrete element and brick walls. The rigid connections modelled contact areas between reinforced concrete elements. Supports were introduced in the location of foundations. Flexibility of supports due to compressibility of soil was not considered.

4. Analysis of model assumptions

Several variants of the building model were calculated in order to test the effects of model assumptions on the results. Variants of modal analyses for the following four partial models were made: 1. The floor plates fixed at the wall lines (Fig. 2a), 2. The floor plates simply supported (Fig. 2b), 3. Floors and walls of rigid connections and fixed wall supports. (Fig. 2c), 4. Floors and walls with mixed supports (Fig. 2d). Joint connections between concrete and masonry elements and rigid connection between concrete elements were assumed in the last scenario.

The use of joints in connections between floors and walls reduced the frequencies of natural vibrations due to the reduction of the building stiffness.
Further analysis was carried out on the model of the building in two variants: with joints in connections between concrete and masonry elements, and with all rigid connections. Natural frequencies and mode shapes were obtained in the modal analysis of the building (Fig. 3).

5. In-situ measurements

Measurements were made during the erection phase, when the building structure was completed. There was no plaster on the walls and no flooring or any additional concrete on the ceilings. Parts of the partition walls were built on the first floor, whilst they were separated from upper ceilings. Basement walls were not covered with ground.

 Forced vibrations were excited in several locations on ceilings, vertical accelerations of free vibrations were measured. Time-dependent accelerations recorded with the use of one of the accelerometers (location P17) are presented in Fig. 4. A total number of 42 tests was carried out in different locations, 3 series in each test. Locations were chosen on consecutive floors of the building. Power spectral densities of accelerations were calculated, next the dominant frequencies were estimated. A spectrum for test P17 is shown in Fig. 5.

The frequencies were determined in order to verify the FEM model. The comparison of calculated and measured values is compiled in Table 1. Dominant amplitudes in calculated mode shapes coincide with locations of the excitation.

6. Conclusion

The presented results are part of larger project which covers analysis of the building coupled with subsoil described by Cam-Clay model. The in-situ measurements in the presented results were used to verify the accuracy of the FEM model of the building overground part. The results of numerical analysis allow to state that the FEM model appropriately represents the structure behaviour. It seems that better results would be received in the rigid connection variant. In the further research the whole object composed of 3 buildings will be modelled. The underground wall supports will be replaced with solid body elements, the subsoil will be modelled too. The influence of subsoil on the building structure will be checked in cases of static analysis. The results will be verified with the in-situ settlement tests.

References

On vibration analysis of wheel composed of ring and plate as an elastic foundation

Stanislaw Noga*1, Roman Bogacz2
1 Faculty of Mechanical Engineering and Aeronautics, Rzeszow University of Technology
Poznaniów Warszawy 12 Street, 35–959 Rzeszow, Poland
e-mail: noga@prz.edu.pl
2 Faculty of Automotive and Construction Machinery Engineering, Warsaw University of Technology
Narbutta 84 Street, 02–524 Warsaw, Poland
e-mail: rbogacz@ippt.gov.pl

Abstract

The paper deals with in–plane flexural vibrations of a wheel composed of a ring and a plate as an elastic foundation. The three–parameter elastic layer is used as a mathematical model of the foundation. The motion equations of the system are derived on the basis of the thin ring theory and Timoshenko beam theory. Analytical models are used to determine the natural frequencies and the normal modes of vibrations of an arbitrarily chosen set of compound annular plates with similar geometry. The eigenvalue problem is formulated and solved using the finite element method for each considered model. The impact of the ring depth on the results of analytical solutions is analysed. The obtained results of calculations are discussed and compared with experimentally achieved data.

Keywords: thin and thick ring, elastic foundation, in–plane flexural vibration, analytical solution

1. Introduction

The vibration theory of circular rings with wheel–plate as an elastic foundation finds applications in several fields of engineering [1,2,4]. The fundamental vibration theory of circular rings is presented in [3]. The theory of thin rings with elastic foundation is used in papers [1,2,4] for the vibration analysis of railway wheels and toothed gears. This paper continues the recent authors’ investigations concerning the vibration of ring with foundation [2]. In this paper the free in–plane flexural vibrations of a circular ring with wheel–plate as a massless three–parameter elastic foundation is discussed using the classic ring theory, and the finite element (FE) formulation. The impact of the ring depth on the results of analytical solutions is analysed. The obtained results of calculations are discussed and compared with experimentally achieved data.

2. Formulation of the problem

Mechanical model of the system under consideration consists of circular ring with wheel–plate as a three–parameter, elastic foundation. Similarly as in paper [2], it is assumed that the centreline of the ring has radius R, and an element of the ring, fixed by angle θ, displaces in the radial and circumferential directions (see Fig. 1a). These directions are denoted as u(θ, t) and ω(θ, t), respectively, where t is time. The coefficients of, k1, k2 and k3 represent the radial and tangential stiffness per length unit, and the ring cross–section angle rotation stiffness modulus [2]. According to results of paper [2], the partial differential equation of motion of the thick ring with elastic foundation takes the form (1). In the case of thin ring on elastic foundation the equation of motion takes form (2). In Eqs. (1–2), E denotes Young’s modulus of elasticity, G is the shear modulus, ρ is the mass density, A is the cross section area and k is the shear correction factor. After using the separation of variables method [3], the so–called frequency equations are achieved and then the values of natural frequencies are determined. During the analysis process, a set of six annular plate models, of the same value of radius R = 0.0875 [m], the geometry as shown in Fig. 1a and different values of rim depth h, are used. In this paper results of investigation for the three representative examples of models are presented and discussed. Technical data of analysed systems have been taken from article [2].

3. Finite element representation

In the elaboration of the FE models of considered systems, the cyclic symmetry feature is used. Each model consists of six homogeneous sectors. Each of these segments is meshed using the six homogeneous sectors. Each of these six homogeneous sectors...
The difference between the reference solutions and the verification of the considered analytical models are discussed. For each analytical model case, the results related to the experimental coefficients of utilized as the reference values. For each analytical model case, the value of the ring substitute mass minimizes the frequency error (Eqn (3)).

<table>
<thead>
<tr>
<th>h [mm]</th>
<th>$k_f$ [N/m²]</th>
<th>$k_p$ [N/m]</th>
<th>$k_s$ [N/m]</th>
<th>$\rho$ [kg/m³]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.004</td>
<td>1.5</td>
<td>6</td>
<td>7.85</td>
<td>11135</td>
</tr>
<tr>
<td>0.008</td>
<td>2.24</td>
<td>6</td>
<td>1.8</td>
<td>9916.1</td>
</tr>
<tr>
<td>0.016</td>
<td>2.23</td>
<td>6</td>
<td>1.6</td>
<td>8393.2</td>
</tr>
</tbody>
</table>

Frequency error $\varepsilon$ [%] (comparison of the thick ring solution with the FE solutions)

| 0.004  | 1.405       | 6           | 0.235       | 9.4           |
| 0.008  | 2.44        | 6           | 0.309       | 9.3           |
| 0.016  | 3.23        | 6           | 0.228       | 8.35          |

Frequency error $\varepsilon$ [%] (comparison of the thin ring solution with the FE solutions)

| 0.004  | 1.405       | 6           | 0.235       | 9.4           |
| 0.008  | 2.44        | 6           | 0.309       | 9.3           |
| 0.016  | 3.23        | 6           | 0.228       | 8.35          |

Natural frequencies of the considered objects $\omega_n$ [Hz] (experimental data)

<table>
<thead>
<tr>
<th>h [mm]</th>
<th>$k_f$ [N/m²]</th>
<th>$k_p$ [N/m]</th>
<th>$k_s$ [N/m]</th>
<th>$\rho$ [kg/m³]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.004</td>
<td>1.5</td>
<td>6</td>
<td>7.85</td>
<td>11135</td>
</tr>
<tr>
<td>0.008</td>
<td>2.24</td>
<td>6</td>
<td>1.8</td>
<td>9916.1</td>
</tr>
<tr>
<td>0.016</td>
<td>2.23</td>
<td>6</td>
<td>1.6</td>
<td>8393.2</td>
</tr>
</tbody>
</table>

Frequency error $\varepsilon$ [%] (comparison of the thin ring solution with the experimental data)

| 0.004  | 1.405       | 6           | 0.235       | 9.4           |
| 0.008  | 2.44        | 6           | 0.309       | 9.3           |
| 0.016  | 3.23        | 6           | 0.228       | 8.35          |

Frequency error $\varepsilon$ [%] (comparison of the thin ring solution with the experimental data)

| 0.004  | 1.405       | 6           | 0.235       | 9.4           |
| 0.008  | 2.44        | 6           | 0.309       | 9.3           |
| 0.016  | 3.23        | 6           | 0.228       | 8.35          |

Figure 1: (a) vibrating system under study; (b) mode shape related to frequency $\omega$ (experimental investigation)

The difference between the reference solutions and the analytical solutions is defined as

$$ e = (\omega - \omega^*)/\omega^* \times 100 \% $$

where $\omega^*$ is the natural frequency from the analytical model, while $\omega$ is the reference natural frequency. First, to determine coefficients of $k_f$, $k_p$, and $k_s$, results from the FE solutions are utilized as the reference values. For each analytical model case, the proper value of $k_f$, $k_p$, and $k_s$, is selected during calculation to minimize the frequency error (Eqn (3)). One of the principal problems, was to determine the value of the ring substitute mass density $\rho$ of analytical models. These values are determined during numerical calculation for thick ring analytical models next they are applied in thin ring analytical models. Results of calculation for the chosen cases of systems are shown in Table 1 (rows 6–12). It is clear that for first two cases, the frequency errors related to analytical models are comparable. Next for two objects related to the first and third discussed system, the experimental test is executed. The LMS measurement set is used in the experimental investigation. The values of excited natural frequencies are shown in Table 1 (rows 14–15) and one mode shape is shown in Fig 1b. These values are compared with the values of natural frequencies of analytical models of considered objects. The appropriate values of the frequency error related to the discussed models are shown in Table 1 (the last four rows). It is worth noting that the achieved values of the frequency error are lower in comparison to the previous obtained case.

5. Conclusions

In the case of presented study, for first two systems, the ratio of $h/R$ is below 0.1 with frequency errors related to both analytical models below 8 [%], which seems to be satisfactory.

References

Thermo-mechanical fatigue of power plant components

Jerzy Okrajni¹, Mariusz Twardawa², Anżelina Marek³*

¹,³ Silesian University of Technology
ul. Krasińskiego 8, 40-019 Katowice, Poland
e-mail: jerzy.okrajni@polsl.pl, anzelina.marek@polsl.pl

² RAFAKO S.A,
ul. Łąkowa 33, 47-400 Racibórz, Poland
e-mail: mariusz.twardawa@rafako.com.pl

Abstract

The paper presents methods for predicting the behaviour of components subjected to the influence of variable temperature and mechanical loading in operating conditions which could be used during reliable fatigue-life predictions for these components depending on the operating parameters and the type of materials they are made from. The research combines an analysis of the operating parameters in the devices under industrial conditions with computer modelling, as well as thermo-mechanical fatigue testing. Selected issues related to heat-exchange conditions as well as the problem of determining the durability characteristics of components under thermo-mechanical fatigue are discussed. The calculations that have been carried out for the devices in question confirm a correlation between the location of the cracks resulting from the conditions of use and the parameters that characterise local strain processes. On the basis of the values of the fatigue parameters determined for different planes at selected points of these components, the locations of the critical planes which are likely to correspond to the areas that suffer the most intensive damage have been determined.

Keywords: power plant components, thermo-mechanical fatigue, critical plane, finite element, stress-strain behaviour

Among the numerous criteria for fatigue life assessment of materials in a complex stress state the critical plane approach [1] is a frequently applied method. The plane is understood as the one in which a parameter assumed as criterial reaches its extreme value. The work proposes its own criterion of thermo-mechanical fatigue life and a method of the fatigue life prediction, which is based on the critical plane approach. A practical example has been presented that shows the method of the fatigue process analysis. One of the power plant components has been taken as such example. The results of operating condition measurements have been described (Fig. 1), which were used to define the boundary conditions for FEM models.

Figure 1. Sample temperature and pressure measurement results for the superheater outlet obtained under industrial conditions

The proper models have been worked out and temperature, strain and stress fields have been determined (Fig. 2).

Figure 2. Temperature distribution over time in the power plant component – overheater header

The models have been validated on the basis of temperature measurements in real components (Fig. 3). The location of the planes, in which stresses and strains in different points of the

Figure 3. The course of temperatures determined on the basis of the model approach for the selected location in comparison with the results of the measurements in industrial conditions

The models have been validated on the basis of temperature measurements in real components (Fig. 3). The location of the planes, in which stresses and strains in different points of the
component (A and B for instance – Fig. 4) were calculated, was defined by the angles $\phi$ and $\nu$ (Fig. 4).

![Figure 4. Location of points A and B. Visualization of the location of the critical plane.](image)

The definition of the criterion parameter has been used in form:

$$ P = \Delta \varepsilon_p \Delta \sigma T_{\text{max}} $$  \hspace{1cm} (1)

$$ P = f(\phi, \nu) $$  \hspace{1cm} (2)

where: $\Delta \varepsilon_p$ – plastic strain range, $\Delta \sigma$ – stress range and $T_{\text{max}}$ – maximal temperature during the fatigue cycle.

The stress versus strain characteristics in different planes have been calculated for different angles $\phi$ and $\nu$ (Fig. 5). The angles which define location of the plane of the maximal value of parameter P have been determined. This value should be used for fatigue life assessment. The fatigue diagrams are necessary in this case. Such diagrams have been worked out in a form, which is shown in Fig. 6. Such diagrams present the results of laboratory material testing. For the calculated value of the P parameter for the chosen point of the components it is possible to assess the number of cycles to crack initiation in defined point.

![Figure 5. Hysteresis loops depending on the values of angles $\nu$ with any value of angle $\phi$ at point A.](image)

![Figure 6. Relationship between the number of cycles to failure and the value of parameter P for X20CrMoV12-1 steel](image)

The Palmgren-Miner rule could be used for changeable in time parameter P.

The calculations that have been carried out for the device in question confirm a correlation between the location of the cracks resulting from the conditions of use and the parameters that characterise local strain processes. This applies to such process parameters as: the plastic strain range in a direction perpendicular to the critical plane, stress range on the critical plane, and the maximum temperature reached during the load cycle. On the basis of the values of these parameters determined for different planes, the location of the plane which is likely to suffer the most intensive damage can be determined. In our case, the critical plane coincides with the plane of the widest range of normal stress and plastic strain. It has also been found that the most intensive process of formation and development of cracks at point A occurs in the plane perpendicular to the axis of the component in question which has been found as the critical plane for this point [4].

These results are a contribution which justifies the advisability of further research combining an analysis of operating parameters in the devices in industrial conditions with research applying computer modelling, as well as thermo-mechanical fatigue testing. The results of thus conducted research should enable the development of methods for predicting the behaviour of the components subjected to the influence of the variable temperature and mechanical loading in operating conditions, and therefore allow reliable fatigue-life predictions for these components depending on the operating parameters and the type of materials they are made of.

### References


Modelling and stiffness control of piezoelectric actuators in an active vibration control system in thin wall machining

Tomasz Okulik, Bartosz Pówalka, Arkadiusz Parus

1,2,3 Faculty of Mechanical Engineering and Mechatronics, West Pomeranian University of Technology, Szczecin
Al. Piastów 19, 70-310 Szczecin, Poland

e-mail: tomasz.okulik@zut.edu.pl, bartosz.powalka@zut.edu.pl, arkadiusz.parus@zut.edu.pl

Abstract

Thin wall machining poses a number of challenges due to a low stiffness of a workpiece. The application of an active vibration eliminator, mounted on the workpiece, is a possible method of tackling the problem. There exists a body of literature reports on active clamping devices used in thin wall machining. The active clamping device is usually fitted with a piezoelectric actuator. The control system regulates the operation of actuators. The authors compared the performance of a system with and without a controller without references to the machining of the workpiece mounted directly on the turntable. The paper presents modelling and simulation stiffness results for piezoelectric actuators used in an active clamping device for thin wall machining. Simulation results are compared with those obtained for the workpiece mounted directly on the turntable. The paper presents a stiffness selection method for an active clamping device depending on the dynamic characteristics of the workpiece.

Keywords: vibrations, active clamping, finite element method, machining of thin-walled pieces

1. Introduction

Thin-walled workpieces are characterized with low stiffness owing to their specific shape with one dimension being much smaller than the other two. Low stiffness of the workpiece makes it susceptible to vibration which can impede machining operations. The machine tool manufacturers and technologists divided thin-walled workpieces into three categories, depending on their height-to-width ratios. Relative to this ratio, different machining strategies are used, including the order passes and machining allowance. Another method of dealing with vibration during machining is to reduce the recommended machining parameters if vibration occurred for the first details in a series.

All the above methods are not efficient in terms of time and material use. Consequently, thin wall machining is not efficient for the purposes of industrial production. The application of an active clamping device may provide a ready solution as it enables control of a displaceable workpiece. Brecher et al. [1] presented an active clamp for a milling machine. The movement of the active clamp was controlled in two directions with piezoelectric actuators. The authors concluded that the application of their active clamp with control system can increase efficiency by 50%, compared to machining conducted without the control system.

Rashid et al. [2] reported an active clamping system for pallet systems. Their clamping system was also controlled with piezoelectric actuators. The authors demonstrated that the clamping system can improve the dynamic characteristics of a system, compared to a system without control.

All the above mentioned authors compared the performance of an active clamping device with and without control system. However, they failed to analyse the machining process of a similar workpiece that would be mounted directly on the turntable of a machine tool.

The aim of the study was to determine stiffness parameters for an active clamping device that could improve machining stability. The paper presents a simulated comparison between machining with the active clamping device and machining while the workpiece is traditionally mounted on the turntable of a milling machine.

2. FEM model

A model of the active clamping device was developed using finite element method. The calculation model was based on the active clamping device designed at the West Pomeranian University of Technology, Szczecin. The workpiece had a height-to-width ratio of 15. It was made of isotropic steel and its structural damping coefficient was 0.1. Piezoelectric actuators were modelled as spring elements connected with the clamping device with stiff rods.

Figure 1: FEM model view: a) piece of standard mounting, b) piece in the active clamping

3. FEM model results

A number of dynamic characteristics for the active clamping device were obtained in numerical simulations. The stiffness of piezoelectric actuators (50, 100, 180, 250, 300, 400 N/µm) varied depending on their available sizes. Figure 2 shows the frequency response function (FRF) for selected stiffness models of piezoelectric actuators compared to the standard mounting on the turntable.
6. Summary and conclusions

The study presents a stiffness selection method for an active clamping device. The paper demonstrated that in order for the active clamping device to fulfill its role its stiffness must be appropriately selected for the dynamic characteristics of the thin-walled workpiece. The example presented above of simulations showed that a relevant stiffness selection of piezoelectric actuators improved machining by 40%, compared to the standard mounting directly on the turntable.

References


4. Simulation model

Piezoelectric actuators built into the clamping system were used to study the machining of the workpiece mounted in a clamp with pre-set stiffness. The actuators are regulated by an external control system using high voltage signal, depending on the workpiece movement parameters. The aim of the control system is to produce such a control signal that the piezoelectric elements would display the properties of a variable stiffness device.

5. Simulation model results

The concept of machining in an active clamping device was validated using numerical simulation. Results demonstrated that piezoelectric actuators can be used in an active clamping system with variable stiffness. Owing to this, the clamping system can be used in machining workpieces with different characteristics. Since the stiffness of the clamping system can be changed during machining operations, the system is suitable to be used for machining workpieces with variable parameters.
Impact of an unsecured excavation on an underground pipeline

Rafał Ossowski¹, Krzysztof Szarf²

¹,² Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland

e-mail: rafał.ossowski@pg.gda.pl¹, krzysztof.szarf@pg.gda.pl²

Abstract

The paper presents a numerical analysis of the impact of an unsecured excavation on an underground pipeline in selected soil conditions. The research was inspired by a real-case failure of a water pipeline which was caused by a neighboring unsecured excavation [1]. The failure was triggered by displacement of a soil mass in the vicinity of the pipeline. The study conducted in the framework of Finite Element Method is focused on analysis of different soil conditions and pipe materials combination and resulting displacements. The FEM parametric study aims at the investigation of the soil model parameter sensitivity on the possible magnitude of horizontal and vertical deformations. The research and analysis can be used in recommendations for engineers performing earthworks in urban areas.

Keywords: pipelines, excavations, quasi-static analysis, soil-structure interaction, FEM modeling

1. Introduction

In urban areas there old pipelines (water, gas, etc.), dating back to the 50s and 60s, still being used are frequent. They are built of quite stiff materials, compared to the modern ones, e.g. in Polish cities there is a large number of cast iron water pipes with bell and spigot joints. They are founded quite shallow, which makes them vulnerable to dynamic loading of heavy trucks etc. Another problem is imprecise documentation of old underground infrastructure (gas lines, district heating, underground power cables and sewer pipes), including those out of service.

In such dense urban areas it is often necessary to perform renovation or reconstruction of certain pipelines which are typically placed directly underneath roads. A standard approach is to secure an excavation, e.g. using sheet-pile walls, trench shield or others. Sometimes only a shallow, up to 2m excavations are needed, and these are often left unprotected [1].

Figure 1: View of water pipeline failure in Gdańsk [1]

Figure 2: Results of water pipeline failure in Gdańsk [1]

2. FEM analysis

The FEM analysis focuses on calculation of soil displacement map surrounding the pipeline, resulting from a shallow unsecured excavation.

The analysis is performed in simplified plane strain conditions, with the use of PLAXIS FEM code. The area is discretised with 15-node triangles and the soil is modeled using Coulomb-Mohr constitutive model. Due to induced large strains this model seems physically adequate (esp. regarding parametric analysis) and applying a more sophisticated model, e.g. Hardening Soil (HS), would not lead to significantly more precise results.

The phenomenon of differential displacements needs to be studied more systematically, therefore the parametric and sensitivity analysis is a proper approach. The resulting vertical and horizontal displacements are calculated for Mohr-Coulomb soil model with different sets of constitutive model parameters (Young’s modulus, Poisson’s ratio, effective internal friction angle and effective cohesion of soil). The magnitude of displacement and loads acting on a pipeline are calculated and compared for different soil conditions.
3. Sample results

The sample analysis refers to a real-life case and presents investigated soil layers. Figure 3 shows a qualitative vector map of displacement field surrounding the pipeline. The maximum magnitude of displacements in this case reaches 70mm. The sample analysis refers to a real-life case and presents investigated soil layers. Figure 3 shows a qualitative vector map of displacement field surrounding the pipeline. The maximum magnitude of displacements in this case reaches 70mm.

![Figure 3: Vector map of total displacements](image)

![Figure 4: Map of horizontal displacements](image)

![Figure 5: Map of vertical displacements](image)

Figures 4 and 5 show the horizontal and vertical components of a displacement field caused by an excavation and traffic load. Horizontal displacements (Fig. 4) are the most intense at the edge of a road, next to the excavation. The total horizontal displacements reaches 50mm—such a displacement can result in a cast iron pipe failure due to the spigot end falling out of the bell (that kind of failure was reported in Gdańsk in 2013 [1]).

In Figure 5 (vertical displacements) large differential settlements occur: more than 65 mm below the road centerline and virtually 0 mm below the edge. Such differences result in high stresses that may cause pipe material failure (fracture). It should be noted that the presented soil displacement values exceed the „safe” levels for stiff and brittle pipes (i.e. concrete or cast iron pipes). In the vicinity of such pipes, especially ones that are aged and weakened (e.g. by corrosion), deformations in the range of centimeters are sufficient to cause a pipeline failure.

4. Conclusions

As previously stated: the results of works made in an unsecured excavation in urban areas can be disastrous for nearby existing infrastructure. An example of such a failure is depicted in Fig. 1 and Fig. 2, more details can be found in [1].

The article focuses on study of soil mass movement in the vicinity of underground pipeline as a result of unsecured excavation and shows the influence of Mohr-Coulomb model parameters on the resulting displacement of soil mass surrounding the pipeline, indicating the most prone conditions to a pipe fracture. The fracture modes are not in the scope of the article, as this problem can be followed in [2,3].

Aside from the parametric analysis a real-life problem case is also described. Some suggestions for engineering earthworks are also given.

References


The influence of presumed border conditions on FEM thermal analysis results based on the example of an LNG tank support saddle

Miroslaw Pajor¹, Jacek Zapłata²

¹,² Faculty of Mechanical Engineering and Mechatronics, West Pomeranian University of Technology, Al. Piastów 19, 70-310 Szczecin, Poland
e-mail: miroslaw.pajor@zut.edu.pl¹, jacek.zaplata@zut.edu.pl²

Abstract

The growing accessibility of CAE environments has made the FEM more egalitarian, calling into question the reliability of some analyses. This paper aims to investigate the influence of some popularly made assumptions on results of FEM thermal analyses on the example of a LNG tank support saddle.

Keywords: thermal FEM analysis, LNG tank, thermal boundary conditions

1. Introduction

The growing accessibility of CAE environments has made the FEM more egalitarian, calling into question the relevance the results of some analyses. Excluding the well-known rules followed to ensure the FEM analysis accurateness, e.g. fine mesh or convergence criteria, one of the most rudimentary requirement is a proper boundary condition formulation. The last requirement often cause problems when thermal analysis of machines or technical structures is to be performed. This paper aims to investigate the influence of some popularly made assumptions on thermal FEM analysis results on the example of a LNG tank support saddle.

In case of a LNG tank foundation the temperature field of the gas tanker hull is of interest since the classification societies (e.g. Polish Ship Register) require that the temperature of construction materials, of a ship, does not exceed allowed service values. The LNG is transported in cryogenic tanks, in a temperature below -162°C. The regulations require confirming by appropriate calculations, that the temperature of the hull not to exceed the allowed temperature borders in any condition, including a leakage of the inner LPG tank [8]. The upper mentioned regulations state, that during those calculations the temperature of surrounding air is to be presumed at level of 5°C, whereas the temperature of the sea water at level of 0°C. Reliability of such calculations is important since the increase of a safety margin causes additional expenditures. The paper considers the influence of usually accepted simplifications on the results of thermal FEM analysis.

2. Presuming a two dimensional temperature field

Two dimensional models are often used in order to decrease the calculation time. Such simplification is allowable if one of the leading dimensions of the concerned body is of the order of magnitude greater than others, or if two opposite border surfaces are isolated. The error made during such a simplification has been shown in Figure 1 on the example of a LNG tank support saddle. The leading dimensions of the tank support saddle were 0,4m x 1,4m x 4,6m. It was assumed that the convection coefficient equal to 5°C/W/m², the surrounding air temperature was equal to -165°C.

Figure 1: The error made as a result of using a 2D model instead of a 3D model [°C]

3. Convection coefficients

3.1. Heat convection coefficients for simple shapes

One of the causes of thermal FEM analysis inaccuracy are imprecisely estimated convection coefficients. The values of these coefficients depend on the surrounding fluid temperature, the physical properties of the surrounding fluid and the shape of the heat exchanging body. One of the most convenient method for estimating the coefficients is the use of literature formulas [5, 6, 7], these defined for some basically shaped surfaces. The accuracy of these estimated coefficients usually does not exceed 15% [7]. As a result, many literature formulas concerning similar geometrical shapes can be found. Figure 2 shows the comparison of the mean Nusselt number for a vertical plane surface in the turbulent flow, estimated on the basis of various literature sources.
The advantage of literature formulas lies in their ease of use and its quickness in comparison to the FEM CFD method. The results are consistent, but the literature methods can only be used for already examined, simple shapes.

3.2. Convection coefficients for unrestricted shapes

The formulas mentioned above are estimated only for basic shapes, e.g. planes, solids of revolution, etc. Though attempts were made to generalize the formulas, still there are no simple empirical ones applicable for complex surfaces [5]. Convection coefficients for complicated surfaces can be estimated by means of FEM CFD methods, or by means of an experiment. These methods are not frequently used, being cost- and time-consuming. Many authors of papers dealing with the subjects of machine temperature analysis or a CNC machine thermal error calculation use a priori chosen values or literature formulas appropriate for basic shapes in order to estimate the coefficients for complex surfaces.

Table 1 shows comparison of convection coefficients calculated on the background of semi adequate literature formulas [7, 9] and FEM CFD analysis. The geometry used to create the comparison is shown in Figure 3. The temperature of selected surfaces was equal 10°C and the surrounding air temperature was equal 0°C.

If the CFD method is taken as a reference, the method [9] seems to be a crude approximation. While studying the Table 1 it is clear that the heat transfer coefficients calculated independently for each wall using literature formulas have values distant from the CFD reference, although the mean values are quite similar.

Figure 4 shows how the variation of heat coefficient acts on the value of the temperature field of the LNG tank support saddle.

4. Conclusions

The paper presents the influence of frequently made simplifications on the result of FEM thermal analyses, on the basis of a LNG tank support foundation analysis. It was shown, that the irrelevant method used for thermal coefficient calculation affects the analysis accuracy. The order of such influence has been reckoned. Other popularly made assumptions have been analysed as well.

References

Self-sustained oscillations of the axisymmetric free jet at low and moderate Reynolds number

Agnieszka Pawłowska1*, Stanislaw Drobnia2*, Piotr Domagała3*

1,2,3 The Faculty of Mechanical Engineering and Computer Science, Częstochowa University of Technology
Dąbrowskiego 69, 42-201 Częstochowa, Poland
e-mail: pawłowska@imc.pcz.czest.pl 1*, drobnia@imc.pcz.czest.pl 2*, piotrd@imc.pcz.czest.pl 3*

Abstract

The paper presents the experimental verification of the self-sustained oscillations found numerically by the team of the Institute of Thermal Machinery. The special design of the experimental rig ensured the relevance between experiment and LES calculations. The measurements of both mean and fluctuating velocity fields confirmed the existence of self-sustained oscillations, however the phenomenon analyzed seems to be Reynolds number dependent.

Keywords: self-sustained oscillations, axisymmetric free jet, shear layer instability

1. Introduction

The research is devoted to the experimental verification of the self-sustained oscillations in axisymmetric jet, which were found by Bogusławski et al. (2013) from LES calculations and received only a limited experimental evidence. These results confirm, that despite a great deal of attention that the round jet received in the last century there remain some significant open questions, and the round jet is still a matter of interest as it is exemplified by the recent experimental work of Mi et al (2013).

The main problems in comparison of experimental and numerical investigations in free jets is the identity of initial conditions, which in the flow considered should be a matter of extreme care. As it was stated by Ball et al. (2012), the round jet is extremely sensitive to inlet conditions, which change not only the near flow field but this influence may propagate even to the far field. Among the most important parameters influencing the axisymmetric jet evolution one may mention the turbulence intensity and the shear layer thickness at the jet exit plane. The experiment set-up with a high contraction ratio of the cubic nozzle ensured both very low turbulence level at the nozzle outlet (Tu << 1%) and thin laminar shear layer characterized by R/θ = 12-18 (where R - nozzle outlet radius, θ - momentum thickness of the shear layer at the nozzle outlet). As it was found by Bogusławski et al. (2013) both these requirements had to be fulfilled in order to obtain the regime of self-sustained oscillations. The measurements were performed with hot wire CTA type 55M DISA and LDA DANTEC 55X System. The spectral and correlation analysis of CTA signals was performed with standard MATLAB software while for LDA signals with randomly spaced samples it was performed by BSA software.

2. Results

Measurements had to be performed for a sufficiently wide range of exit shear layer thickness, the thickness of exit shear layer was varied by the cylindrical extensions with various lengths, measurements were performed for three values of Re number equal 5x10^3, 10x10^3 and 20x10^3 respectively.

For each value of Re four cylindrical extension tubes of various lengths were applied, the summary of exit shear layers are presented at Table 1, where R/ θ is the non-dimensional values are extremely low, that allows to expect that all preliminary conditions outlined by Bogusławski et al. (2013) are fully met by experimental test rig. The sample evolution of mean velocity U and turbulence intensity Tu along the jet axis are presented at Fig. 1 for all values of exit shear layer thickness.

The data presented at the Fig.1a reveal a typical jet behaviour, the thicker is the shear layer at the exit the longer becomes the potential core, while the Tu evolution shown at Fig.1b confirm another regular behaviour related to the existence of the “plateau”, which is related to the presence of coherent structures, as it was shown by Crow and Champagne (1971). One may conclude therefore, that the increase of exit shear layer thickness reduces the jet spreading angle and increases the length of jet potential core because of less favourable conditions for the development of coherent structures. The results of spectral analysis (not shown here) revealed the peaks corresponding to vortical structures developing in the initial region of the jet.

Table 1: Parameters of exit shear layer

<table>
<thead>
<tr>
<th>Re</th>
<th>R/θ</th>
<th>H</th>
<th>Tu[%]</th>
<th></th>
<th>Re</th>
<th>R/θ</th>
<th>H</th>
<th>Tu[%]</th>
<th></th>
<th>Re</th>
<th>R/θ</th>
<th>H</th>
<th>Tu[%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>5x10^3</td>
<td>35.7</td>
<td>2.591</td>
<td>0.16</td>
<td></td>
<td>54.3</td>
<td>2.567</td>
<td>0.13</td>
<td></td>
<td>80.9</td>
<td>2.617</td>
<td>0.05</td>
<td></td>
<td>2x10^4</td>
</tr>
<tr>
<td>22.3</td>
<td>2.486</td>
<td>0.21</td>
<td></td>
<td>31.3</td>
<td>2.533</td>
<td>0.14</td>
<td></td>
<td>39.0</td>
<td>2.700</td>
<td>0.07</td>
<td></td>
<td>31.0</td>
<td>2.541</td>
</tr>
<tr>
<td>15.3</td>
<td>2.555</td>
<td>0.22</td>
<td></td>
<td>20.1</td>
<td>2.540</td>
<td>0.14</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>80.9</td>
<td>2.617</td>
<td>0.05</td>
<td></td>
<td>2x10^4</td>
<td>39.0</td>
<td>2.700</td>
<td>0.07</td>
<td></td>
<td>31.0</td>
<td>2.541</td>
<td>0.07</td>
<td></td>
<td>22.6</td>
</tr>
</tbody>
</table>

*The investigations presented in this paper have been obtained with funding from Polish National Centre of Science within the grant DEC 2011/03/B/ST8/06401 and Statutory Funds BS/PB-1-103-3010/2011/P.
In particular it was found that the amplitudes of the dominant mode was much stronger than for other harmonics, that was typical for externally stimulated jets.

Since the jet was not stimulated in the present experiment so the peaks obtained must be a result of self-sustained oscillations caused by the development of coherent structures.

![Figure 1: Mean a) and fluctuating b) velocity at the jet axis for Re = 10 000](image)

The amplitudes of dominant peaks from all spectra recorded for various Re and various ratios of $R/\theta$ were presented at Fig. 2 as a relation between Strouhal number based on jet exit diameter and the relative thickness of exit shear layer.

Furthermore, the results of LES calculations taken from Ref. [3] were also presented here. The relevance between LES and present experiment is qualitative only but the most important is the same tendency that was observed here and which was also present in the previous research reported in Ref. [3]. For $Re = 10^4$ and $2 \times 10^4$ one may observe that for the certain value of $R/\theta$ the constant $St_D$ is obtained that results from the presence of self-sustained oscillations.

![Figure 2: Non-dimensional frequency (Stouhjal number) of the most amplified mode versus shear layer thickness obtained from LES calculations [3] versus present experiment](image)

The results obtained with the present experimental set-up confirm the existence of self-sustained oscillations, however the phenomenon analyzed seems to be Reynolds number dependent. In particular, for the lowest value of $Re$ the jet behavior resembles the results of linear theory calculations. The increase of $Re$ leads to the behavior suggested by LES simulations, which are characteristic for large amplitude self–excited flow oscillations. It should also be noticed that the results obtained in the present experiment appear to be very similar to those typical for externally forced jets, further studies were mainly based on comparison with LES numerical investigations and concentrated on explaining this unusual jet behavior. In particular it was found, that these self-sustained oscillations, despite that they are growing locally in time up to high amplitudes limited by non-linear interactions, being in this feature similar to the self-sustained absolutely unstable mode present in hot jets, are of certainly convective character originating from the classical Kelvin-Helmholtz instability. Since the self-sustained regime seems to be a new finding as a consequence it required to explain how this regime reacts to density variations in heated jets and how this self-sustained mode interacts with absolutely unstable mode, when the density ratio is lower than the critical one for an absolutely unstable regime.

3. References


Numerical prediction of fusion zone and heat affected zone in Yb:YAG laser heating process with experimental verification

Wiesława Piekarska1, Marcin Kubiak2

1 Institute of Mechanics and Machine Design Foundations, Częstochowa University of Technology
Dąbrowskiego 73, 42-201 Częstochowa, Poland
e-mail: piekarska@imipkm.pcz.pl, kubiak@imipkm.pcz.pl

Abstract

The paper presents mathematical and numerical modelling of temperature field during Yb:YAG laser heating of sheets made of S355 steel with the motion of liquid steel in the fusion zone taken into account. Interpolation models for a precise description of Yb:YAG laser power intensity distribution are developed. Laser power distribution and the caustics are determined on the basis of the geostatistical Kriging method. The input data for the model are obtained from experimental research made on TruDisk 12002 laser. Temperature field and melted material velocity field in the fusion zone are obtained from the numerical solution of continuum mechanics equations using Chorin projection method and a finite volume method. In order to verify the correctness of developed model an experiment is performed. Characteristic zones of the cross section of heated element are compared to a numerically predicted fusion zone and a heat affected zone.

Keywords: Yb:YAG laser heating, laser heating, thermal phenomena, fluid flow, numerical modelling

1. Introduction

One of the most intensively studied lasers are diode pumped Yb:YAG lasers. Issues concerning the use of this type of solid state lasers in the industry are under particular investigations in the field of experimental research as well as theoretical study [1].

The major factor determining the temperature field in the laser beam processing is the beam power and its distribution in heated element. Changing focal length of focusing lens in the optical system determines the laser spot diameter as well as laser beam power intensity distribution. The effects of non-Gaussian beam pumping and no desirable thermal lensing effects are usually omitted in the analysis. Only a specific laser beam spot diameter is adopted in analytical models and the beam focusing as a technological parameter influencing the laser beam intensity distribution is neglected. Consequently, the laser beam intensity distribution models assumed in numerical analysis significantly differ from real Yb:YAG laser profile, obtained through experimental research [2].

2. Mathematical model

Differential governing equations used in the analysis consist of mass, momentum and energy conservation equations.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho v_i) = 0 ,
\]

\[
\frac{\partial (\rho v_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho v_j v_i) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial v_i}{\partial x_j} \right) + \rho g \beta_T \left( T - T_{ref} \right) - \frac{\mu}{\rho K} v_i ,
\]

\[
\frac{\partial}{\partial x_i} \left( \lambda \frac{\partial T}{\partial x_i} \right) = C_v \left( \frac{\partial T}{\partial t} + \rho \frac{\partial T}{\partial x_i} \right) - Q ,
\]

where \( \rho \) is a density [kg/m³], \( g \) is acceleration of gravity, \( \beta_T \) is a volume expansion coefficient due to heating [1/K], \( T_{ref} \) is a reference temperature [K], \( \mu \) is a dynamic viscosity [kg/ms], \( K \) is porous medium permeability described by Carman–Kozeny equation, \( T = T(x,t) \) is temperature [K] at a point \( x \), \( v \) is a velocity vector [m/s], \( k = \lambda(T) \) is a thermal conductivity, \( C_v(T) \) is an effective heat capacity with latent heat of fusion and evaporation taken into account [3].

Initial conditions and boundary conditions complete governing equations. Equation (2) is completed by initial condition \( t = 0 : v = 0 \) and boundary conditions implemented at the welding pool boundary determined by solidus temperature \( T_{sol} = T_s \), described as follows:

\[
\Gamma : \ n \cdot \frac{\partial y}{\partial n} = \frac{\partial y}{\partial t} , \quad v = 0 ,
\]

\[
\Gamma : \ n \cdot \frac{\partial y}{\partial n} = \mu \frac{\partial y}{\partial t} \frac{\partial T}{\partial s} ,
\]
where \( \tau_s \) is Marangoni shear stress in the direction tangent to the surface, \( \gamma \) is surface tension coefficient.

Equation (3) is completed by initial condition:
\[ t = 0 : T = T_0 \]
and boundary conditions, taking into account heat loss due to convection, radiation and evaporation:
\[ \Gamma : -k \frac{\partial T}{\partial n} = \alpha(T - T_0) + e\sigma(T^{4} - T_0^{4}) - q_e + q_s, \]
where \( \alpha \) is convective coefficient \([W/m^2K]\), \( e \) is radiation coefficient, and \( \sigma \) is Stefan-Boltzmann constant. Element \( q_e \) is the heat flux towards the top surface of the welded element \((z=0)\) in the source activity zone, while \( q_s \) represents heat loss due to material evaporation in area where \( T=2T_n \), \( \Gamma \) is a boundary of analyzed domain.

The governing equations are numerically solved using a projection method with a finite volume method [4].

Interpolation algorithms are used to describe of heat source power distribution, allowing a precise description of Yb:YAG laser power intensity. Models elaborated on the basis of ordinary Kriging method [5] in the form of point Kriging take into account the real laser power distribution obtained in experimental research made on laser welding station equipped with TruDisk 12002 laser.

Kriging interpolation at a point \((x, y)\) is a linear combination of observations in basic points (the real power distribution). The estimate is a function of the weighted average:
\[ \tilde{f}(x, y) = \sum w_i f(x_i, y_i) \]  
where \( w_i \) are weight coefficients, calculated on the basis of Kriging system of equations, assigned to particular observations. \( f(x_i, y_i) \) is the real value of the function (variable) at the estimated point. \( n \) is the number of sampling points that are considered in estimating of the variable within the circle of radius \( r \) from the estimated point.

3. Results and discussion

Numerical algorithms are implemented into a computer solver using ObjectPascal programming language. Computer simulations of the Yb:YAG laser heating process are performed for S355 steel sheets with dimensions 250mm in length, 50mm in width, with a thickness of 5mm. The analyzed domain is discretized by a staggered grid with the spatial step set to 0.1 mm. The numerical analysis is made using interpolated heat source model with laser beam power \( Q = 900 \) W, laser head movement speed \( v = 3 \) m/min with the focusing point on the top surface of heated element \((z=0)\). Basic parameters of the laser beam used in the model are obtained using Prometec UFF100 beam analyzer, a diagnostic device for measuring high power lasers profile.

Figure 2 shows heat source power distribution obtained by Kriging method for interpolation grid step \( \Delta h = 0.02 \) mm. Figure 3 illustrates the temperature distribution at the top surface and in cross section (for \( x=3 \) mm) of laser heated sheet. In Fig. 3 a comparison of predicted characteristic zones of heated element with the macroscopic picture of the cross section of heated element is presented.

4. Conclusions

Application of a Kriging algorithm allows to use experimental studies in numerical analysis and allows for a precise reproduction of the real thermal load depending on the laser profile obtained for specified industrial laser.

It can be observed that numerically estimated fusion zone and heat affected zone well agree with experimentally obtained macroscopic picture of the cross-section of a laser welded joint.

The performed comparison of the heat source power distribution model with experimental data as well as characteristic zones of heated element corroborates the validity of performed theoretical assumptions and usability of model in practical applications.

References

Stress redistribution at the support of transversely loaded sandwich panel

Zbigniew Pozorski¹, Jolanta Pozorska²

¹ Institute of Structural Engineering, Poznan University of Technology
Piotrowo 5, 60-963 Poznan, Poland
e-mail: zbigniew.pozorski@put.poznan.pl

² Institute of Mathematics, Czestochowa University of Technology
Armii Krajowej 21, 42-200 Czestochowa, Poland
e-mail: jolanta.pozorska@im.pcz.pl

Abstract

The paper concerns the problem of static structural behaviour of sandwich panels. The local stress concentration and distribution at the supports are discussed. The 3-D numerical model with the definition of damage initiation and evolution was applied. Different failure modes are taken into account. The numerical results are compared with real experiments and engineering simplifications. The assumption concerning uniformity of stress distribution is verified. Known from the literature methods of evaluating of the capacity of the sandwich panel are discussed.

Keywords: composites, sandwich panels, soft core, finite element method, structural behavior, indentation

1. Introduction

The paper considered sandwich structures, which consist of two thin external steel faces and thick and soft core. The sandwich elements are very attractive because of high load-bearing capacity at low self-weight and excellent thermal insulation. Unfortunately, in sandwich structures may occur various failure mechanisms. Type of the failure depends on many geometrical and mechanical parameters, as also on boundary conditions [2]. The following failure mechanism can be distinguished: face yielding, global and local instability, debonding (usually caused by impact loads), shear and indentation of the core.

An important failure mode of sandwich panel, which can occur at the point loads, is local indentation. Crushing of the core under the acting force causes such damage. Depending on the form of force excitation, flexural stiffness of sandwich a face and stiffness of a core, applied force spreads at different area. The indentation loads were studied for various materials [6,9] and the great significance of an indenter shape was reported [8]. Experimental studies of the indentation phenomenon are dominating, because of the complexity of the problem [4]. Theoretical models of indentation were started with sheet treated as elastic beam located on the ideal elastic foundation [5]. Currently, there are more complex models, which assume plastic yielding of the core [7], rigid and ideally plastic behavior of core and face material or elastic deformation of the faces and compressive yield of the core [1].

This paper discusses the engineering problem of stress redistribution at supports of sandwich panel. Sandwich panels applied in civil engineering are installed to the substructure (purlins, side rails). Interaction between substructure and the panel induces compression of the sandwich core. The stress concentration can be particularly important in the case of multi-span systems loaded by temperature gradient. From a practical point of view, it is essential to spread out the impact of the support on a relatively large area of the panel. The paper takes up the problem of estimation of the stress redistribution at the supports. The influence of geometrical and mechanical parameters of the system on the redistribution level are discussed, too.

2. Problem formulation

Consider the structure presented in Fig. 1. The relatively short panel is subjected to one-line loading to provide high stress concentration at the end support. This system corresponds to the assumptions of the standard [3].

![Figure 1: The system for determination of stress redistribution at the support](image)

The reaction of the support can be calculated using the relation:

\[ F_R = \frac{L_2}{L_1+L_2} F. \]  

Looking for a parameter describing the distribution of the support reaction, we can define the support capacity \( F_R \) using the distribution parameter \( k \):

\[ F_R = b(L_s + 0.5ke) f_{cc}, \]  

where \( b, L_s, e \) and \( f_{cc} \) denote width of the panel, width of the end support, distance between centroids of faces and compression strength of the core, respectively. The parameter \( k \) determines the effective width of the support (and thus the support reaction redistribution) related to the mid-plane of the panel.

439
The aim of the paper is the assessment of stress distribution in the core and verification of the assumption concerning the uniformity of the distribution. The problem is analysed numerically. The results are compared with real experiments and interpretation with respect to Eqn (2) is presented.

3. Numerical model

In numerical simulations the parameters of the structures correspond to the previously carried out experiments. The model was prepared in the ABAQUS system. Only 3-D models were applied. Sandwich panel with the length 2.20 m is located on two supports ($L_1=0.34$ m, $L_2=1.36$ m). Width of the left support is 0.10 m, whereas the width of the right support is 0.04 m. The width of the loading element is 0.08 m. Core compression at the right support is analyzed. The total depth of the panel is 98.43 mm. The thickness of each of the faces is $t_1 = t_2 = 0.471$ mm, hence $e=97.959$ mm. Steel, flat facings were assumed as elastic-plastic material. The elasticity modulus $E = 195$ GPa and Poisson’s ratio $\nu = 0.3$ were used. The actual relationship between stress and strain was introduced. The yield strength was 360 MPa, and the tensile strength reached 436 MPa. Facings were modelled using four node, doubly curved, thin or thick shell, reduced integration, hourglass control, finite membrane strain elements S4R. The core of the panel was modelled using eight node linear brick elements C3D8R. The core was considered as isotropic elastic material with $E_c = 8.61$ MPa, $\nu_c = 0.02$ or as elastic-plastic material with actual yield stress - plastic strain relation.

Between the core and the faces, the interface layer of the thickness 0.5 mm was introduced with definition of damage initiation and propagation. The interface was modeled using COH3D8 8-node, 3D cohesive elements. The uncoupled elasticity law for cohesive material was used. The quadratic nominal stress criteria of damage initiation and displacement type, linear softening damage evolution was defined. Interaction between all parts was assumed as TIE type, which makes equal displacements of nodes. Between the right support and the sandwich structure surface-to-surface contact was examined.

4. Discussion of the results

An exemplary numerical solution is presented in Fig. 2. The figure shows normal stress $\sigma$ (perpendicular to the faces) in the core part. Of course stress concentration is observed in the vicinity of the support and the area of the force application. The conducted simulations show that local stress concentrations can be higher at the area of force application than at the right support.

An even greater problem is with the shear stress that also in the case of relatively thick core are higher than the normal stress. As a result, simulation of the end support compression leads to other forms of failure than core crushing (generally referred to compressive strain 10%). Therefore, the identification of the distribution parameter $k$ on the basis of the maximum load measured in the test seems to be wrong, or at least very safe. For example, for local compressive normal stress at the support 120,9 kPa extreme shear stress in the core is 154 kPa, and the change in thickness of the core is only about 1%. The analysis of the stress distribution in the core shows that the average value of the compressive stress at the support, at half the height of the core is approximately 40 kPa.

5. Conclusions

A proposed numerical model is useful in the analysis of the problem of stress redistribution at the supports of sandwich panels. The 3-D model allows to assess the level of stress concentration and. Introduced interface layers and constitutive relations enable observation of the different mechanisms of destruction.

The method of evaluating the capacity of the sandwich panel at the support proposed in the literature and discussed in the paper is inappropriate. The distribution parameter $k$ determined according to Eqn (2) does not reflect adequately the phenomena occurring at the support. Further simulations and analysis should be aimed to find the appropriate test methods of sandwich panels.

**References**


Three-point bending test of sandwich beams supporting the GFRP footbridge design process - validation analysis

Łukasz Pyrzowski 1, Bartosz Sobczyk 2, Wojciech Witkowski 3, Jacek Chróścielewski 4*

1,2,3,4 Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
E-mail: lpyrzow@pg.gda.pl 1, barsobcz@pg.gda.pl 2, wojwit@pg.gda.pl 3, jchrost@pg.gda.pl 4

Abstract

Selected aspects, concerning architectural, material and construction design issues for pedestrian footbridge made of GFRP composite materials are elaborated in this paper. The analysis is focused on validation tests, which are particularly important because of the advanced technology and materials that are used for this innovative bridge. The considered footbridge is a sandwich-type shell structure comprising of PET foam core and outer skins made of glass fibre reinforced polymer laminate. A proper connection of foam core and laminate skins is crucial, regarding its load bearing capacity. The aim of this paper is to compare experimental and Finite Element Analysis (FEA) results of sandwich beams subjected to bending loading in three point bending tests, which are afterwards used to validate the stiffness properties of the full structure.

Keywords: composite footbridge, sandwich structure, GFRP laminate, PET foam core, validation, bending test

1. Introduction

The general subject of the study is to elaborate architectural, material and construction design aspects, concerning the design process of pedestrian footbridge made of composite materials. A typical span has an U-shape sandwich type cross section comprising of polyethylene terephthalate (PET) foam core and skins made of glass fibre reinforced polymer (GFRP) laminate. The simply supported 14 m long spans are intended to be applied for instance over two lane roadways. The design concept is presented in Figure 1. More detailed information about the project can be found in [1].

Figure 1: Side view of the composite footbridge

The standard design process includes the development of concepts and material selections, identification of material properties, numerical simulations, strength calculations and serviceability analyses. In the case of innovative structures, especially made of non-classical materials, which are not included in the existing design codes [2], it is necessary to perform additional validating analyses. Material parameters identification as well as experimental and numerical investigations are also required during the process of the innovative bridge design. Some selected above mentioned aspects are described in details in the paper. Comparison of the experimental and FEA results of three-point bending tests carried out on sandwich beams is the most important aspect of this paper.

2. Experimental investigations

2.1. Description of specimens

Sandwich beams (laminate-core-laminate) with 90x74x1220 mm cross sectional dimensions (Figure 2) are utilised during validation tests. The main components of GFRP laminate are vinylester resin and three layers of glass fibre knitted fabrics. The core is made of 100 kg/m³ PET foam. The samples, such as the intended footbridge, are formed using the infusion technology.

Figure 2: Geometry of composite beam

2.2. Experimental tests

The sandwich specimens are subjected to three-point bending tests. The test setup is shown in Figure 3. The experiments are carried out by the Zwick/Roell Z400 testing machine using the displacement control technique with the speed of the cross-head of 2 mm/min. The vertical displacement of the beam is measured with inductive sensor. Longitudinal skin strains are measured using resistance strain gauges, during the experiment.

*The study has been supported by the National Centre for Research and Development, Poland, as a part of a research project No. PBS1/B2/6/2013, realized in the period 2013-2015. B. Sobczyk is supported under Gdańsk University of Technology (Poland), Faculty of Civil and Environmental Engineering, Young Scientist Support Programme. The support is gratefully acknowledged. Abaqus calculations were carried out at the Academic Computer Centre in Gdańsk.
3. Numerical model

Some numerical simulations of three-point bending test of sandwich beams (described in chapter 2) are carried out in order to assess whether the design assumptions, concerning material models selection and material constant experimental determination are correct. The discrete models (see Figure 4) are built in Abaqus 6.14-2 environment. Two geometry variants are considered. Symmetry is introduced in both cases with respect to the beam width (Z-symmetry), in order to reduce the numerical effort of the task. The first model represents pure laminate-foam-laminate structure. Additional resin columns are included in the second one, which are a consequence of the infusion technology. Their location was not symmetric regarding the beam length in the tested specimen. Their height is the same as in the foam core. The distance between adjacent columns is equal to 60mm in the beam length and width direction. Two columns are observed in the specimen regarding the beam full width, therefore only one per width is included in the second numerical model.

The laminate skins are modelled with S4 elements. Equivalent single layer technique is utilised to represent behaviour of multi-layered medium. C3D8 finite elements represent the foam. We assume that the cross-head is rigid, while supports are made of regular steel (also C3D8 elements). Additional resin columns, used in 2nd model variant, are built with truss T3D2 finite elements. The first model contain 20325 and the second one is built of 20365 nodes.

Plane-stress orthotropic material law is used to represent behaviour of lamina stiffness and strength properties. The material constants are derived on the basis of experiments [3]. Isotropic constitutive relation is used for foam material, basing on the data provided by the material distributor. Crushable foam behaviour with hardening is included for the core description, in which the material constants are derived by means of experiments. The laminate skins are bonded to the foam. A friction contact is assumed between: the cross-head and laminate upper skin and between laminate lower skin and rigid supports interfaces. The resin columns are included in the second analysis variant using the embedded region Abaqus technique. Isotropic material model is used to represent their behaviour.

Geometrical and material non-linearities are included in the static calculations in order to describe the structure behaviour. The rigid cross-head is moving in the downward direction by 24mm during two FEA. Cross head rotations and displacements in other directions are restricted. All translational degrees of freedom located on the lower surface of the beam supporting elements are blocked.

4. Comparison of results

Figure 5 presents comparison of total force versus extreme beam vertical displacement relations for the experiment and two numerical models.

Both numerical models are able to describe the linear response of the sandwich beam properly. Because laminate skins failure is not considered here only the foam destruction process is initiated during numerical simulations. Therefore, the load carried by beam don’t decrease rapidly in FEA. It is also noticeable that, when additional resin columns are included, the total beam load capacity can be estimated more precisely.

5. Conclusions

The design process of an innovative footbridge is very complex. Many additional calculations, simulations and experiments are required to be carried out. Two finite element simulations of the sandwich beam three-point bending test revealed that the design variables as material constants and constitutive relations are correctly determined. The FEA also revealed that the numerical approach used here also allow to describe a proper behaviour of composite structure. It can be concluded that it is possible to design a full scale footbridge with use of information determined from the presented research.

References


Numerical analysis of scaffolding stands with defects

Aleskander Robak¹, Ewa Blazik-Borowa², Jarosław Bęc³

¹,²,³ Faculty of Civil Engineering and Architecture, Lublin University of Technology
Nadbystrzycka 40, 20-618 Lublin, Poland

Abstract

Methods of numerical modelling and damage analysis of the main structural members of facade scaffolding systems, i.e. stands, were presented in this paper. Practically unavoidable structural and material defects result in reduction of the structural strength. Numerical analysis preceded by experiments allows description of structural members’ performance in details.

Keywords: nonlinear static analysis, scaffolding, steel pipe, optical 3D deformation analysis, structural defects

1. Introduction

Analysis of defects influence on static performance of scaffolding stands with a deflected part, was scanned with the use of the 3D scanner. The obtained geometry represents the outer surface of the real pipe, and the ones obtained from the scan, equal. The second model was used as the thickness of the element in the scanned pipe and in the perfect pipe, was connected to create similar triangular plate elements with pairs constituting near planar rectangular areas.

Two ways of introduction of the damaged area into the model were used. The first one is based on the change in the area of the elements in the scanned pipe model in reference to the perfect pipe respective areas. Ratios of the respective areas from two models were calculated. The thickness of each element of the damaged pipe has been adjusted to make the volume of each pair of elements, i.e. constituting the perfect pipe, and the ones obtained from the scan, equal. The second method is based on the displacements of the nodes from the perfect surface. The more value of displacement is, the more thickness of the element is changed.

Numerical static nonlinear analysis was performed on another model which mesh is made quite independently of the mesh of the scanned pipe. The scanned surface was imported to the pre-processor of Autodesk Simulation software and scaled in two directions to obtain the pipe middle surface.

The meshes of the scan coming from 3D scanning, i.e. of the deflected pipe and of the perfect pipe are more or less regular. The areas of the elements are of the same order in both models and independent of the location of the damage. However, when calculating the FEM model, the mesh should be much denser at the locations closer to the damage. This generated the problem of projection of the elements parameters between models. So each element thickness in the FEM model was adopted as the thickness of the element in the scanned pipe whose centre is the closest to the centre of the respective element in the FEM model.

2. Description of the analytical model

In the first stage, the analysed part of the scaffolding stand pipe with a defect, was scanned with the use of the 3D scanner. The obtained geometry represents the outer surface of the real pipe with the deflected part where the pipe had been hit with a tool. The 3D scan results with coordinates of points located on levels with 1 mm distance from each other. Total length of 10 cm pipe has been scanned and this results with 101 levels of points.

On the basis of the top and bottom parts of the deflected pipe, the reference model of the undamaged pipe was generated. First, the centre line of the pipe was calculated as a straight line between centres of the top and bottom circles. Then the radius of the perfect pipe was calculated on the basis of ten levels of radii at each end of the scanned part of the pipe (where there is no significant influence of the damage on the geometry).

Each point of the model coming from 3D scanning was radially projected on the surface of the perfect pipe resulting with the respective point of the perfect pipe. Points in each model, i.e. in the scanned pipe and in the perfect pipe, was connected to create similar triangular plate elements with pairs constituting near planar rectangular areas.

Shell elements were used in the analysis. The FEM model with the shape following 3D scan and with the automatically generated mesh was subjected to further analysis. The mesh was additionally refined at the points where stress concentration is expected. The 10 cm model was symmetrically elongated to the total length of 40 cm to match the length of the specimen used in the reference measurements. The mesh of the numerical model used in static nonlinear analysis was presented in Fig. 1.
Figure 1: Numerical model

Non-linear static analysis with use of Autodesk Simulation Multiphysics 2013 was performed with consideration of material and geometrical nonlinearities. In order to reproduce the experiment as accurate as possible, the load was applied with displacement control at one of the specimen ends. Figure 2 shows exemplary maps of strains obtained with the numerical analysis (left) and registered in the experiment with use of ARAMIS system by optical 3D deformation analysis (right).

Figure 2: Exemplary map of deformations

There have been marked two corresponding points at each of the pictures. These pairs of points have been used to calculate longitudinal strains at the part containing the defect.

Relation of the applied force to strain obtained on the basis of relative displacements in the points shown above is presented in Fig. 3. The results coming from experiment and the numerical simulations were juxtaposed. The line marked as “Model I” represents the results obtained in the numerical simulation with no damage included. Then additional effects coming from the real damage were introduced in the subsequent models. In the model II, only the local defect (pipe surface deformation) was introduced. As expected, this effect has bridged the gap between results from experiment and the numerical simulation most of all. In the model III, there has been additionally introduced the whole pipe deflection which had emerged during the generation of the damage. In the models IV and V there were included local changes of the pipe thickness, as described above in the point 2. In the model VI, the material characteristics inside the damage area were changed on the basis of the calculated strain. As it can be seen, the consecutive efforts closed the gaps between the respective results. However, only in the initial part of the graph, where we can see linear relations, the lines matched exactly. The observed inadequacy in other ranges comes from some simplifications introduced in calculations, such as the applied material model with linear hardening, as well as from the errors or inaccuracy in the tests made with use of the ARAMIS system.

Figure 3: Relation of axial force to strain

4. Conclusions

After the introduction of changes in thickness and material properties for shell elements in the area, the strain values observed at maximum forces in numerical analysis have matched the ones obtained in experiment.

The analysis presented in this paper allows finding the matching characteristics of the material and geometry for the shell model of scaffolding stands.

Further analysis is planned to transfer the results from shell models to the simplified beam models of the whole scaffolding frame structure with equivalent material characteristics considering presence of defects in stands.

References

Modeling of aluminum extrusion process based on Bodner-Partom model

Grażyna Rzyńska¹, Andrzej Skrzat²
¹,² Faculty of Mechanical Engineering and Aeronautics, Rzeszów University of Technology
Al. Powstańców Warszawy 12, 35-959 Rzeszów, Poland
e-mail: grar@prz.edu.pl ¹, askrzat@prz.edu.pl ²

Abstract

Numerical results of aluminum 1100 extrusion process for a tool velocity equal 10 m/s are presented. Viscous and plastic properties of the extruded materials are described on the basis of numerical simulations of a tensile test in which the Bodner-Partom material model is applied. Numerical results are compared with the experimental ones. Engineering applications of the considered problem concern the use of the extrusion technology in the design of the impact energy-absorbing device.

Keywords: aluminium 1100, Bodner-Partom, extrusion

1. Introduction

Competition and intensification in the production require developing new materials of required mechanical properties. Enhancements in forming processes, as well as the search for new unconventional use of this technology provides the need for the correct description of the behavior of metallic materials at very high strain rates, often up to 5000 s⁻¹. Some ability to describe material behaviour under high strain rates can be obtained by application of mathematical models such as: Bodner-Partom (1975) [1], Zerilli - Armstrong (1987) [2] or Johnson - Cook (1983) [3]. Their use is associated with certain assumptions and limitations. There is necessary to derive the coefficients and constants for each material model which usually is not straightforward. Numerical results of 1100 type aluminum extrusion at 30 km/h tool speed are presented in this paper. As a practical engineering application, the extrusion technology in the design of impact energy-absorbing devices is considered.

2. Bodner-Partom material model

The Bodner-Partom material model (1975) is formulated as a set of constitutive equations representing elastic-viscoplastic strain-hardening behavior for large deformations and arbitrary loading. An essential feature of the formulation is that the total deformation consists of the elastic and inelastic components, which are functions of the material parameters and state variables at all stages of loading and unloading. The Bodner-Partom material model allows to take simultaneously into consideration elastic and plastic effects, isotropic and kinematic hardening, visco-plasticity, creep and relaxation for a wide range of temperature. The Bodner-Partom material model is defined by the following equations:

\[ \sigma = \frac{\sigma_y^e}{J_2} \exp \left( -\left( \frac{Z + Z^\\rho}{3J_2} \right)^n \right). \]  

(4)

In Eqn (4) \(\sigma_y^e\) and \(n\) are B-P material constants, \(J_2\) is the second invariant of deviatoric stress. State variable \(Z = Z + Z^\\rho\) represents the resistance of material to inelastic deformations - both isotropic \(Z\) and directional \(Z^\\rho\). The evolution of \(Z\) is defined as

\[ Z^I = m_i(Z_i - Z^I)\sigma_y^e, \]

(5)

Parameter \(Z^\\rho\) depends on tensor quantity \(\beta_y\)

\[ Z^\\rho = \frac{\sigma_y^e}{\beta_y}. \]

(6)

where

\[ \beta_y = m_i(Z_i - Z^\\rho)\sigma_y^e, \]

(7)

Here: \(m_i, m_{i1}, A_i, A_{i2}, Z_i, Z_r, r_1, r_2\) are B-P material constants described below.

For inelastic strain rate derived from (3) the elastic stress rate is determined from the generalized Hooke’s law

\[ \dot{\sigma}_y^{(v)} = C_{\dot{\varepsilon}} \left( \dot{\varepsilon}_y - \varepsilon_y^{(v)} \right). \]

(8)

Bodner-Partom material parameters are given below:

\( E\) [MPa] - elastic modulus;
\( v\) [-] - Poisson’s ratio;
\( D_0\) [s⁻¹] - limiting shear-strain rate;
\( Z_0\) [MPa] - Initial value of isotropic hardening variable;
\( Z_i\) [MPa] - limiting value for isotropic hardening;
\( Z_1\) [MPa] - limiting value for kinematic hardening;
\( m\) [MPa] - hardening rate coefficient (isotropic);
\( m_{i1}\) [MPa] - hardening rate coefficient (kinematic);
\( n\) [-] - strain rate sensitivity parameter;
\( A_i\) [s⁻¹] - recovery coefficient for isotropic hardening;
\( A_{i2}\) [s⁻¹] - recovery coefficient for kinematic hardening;
\( r_1\) [-] - recovery exponent (isotropic hardening);
\( r_2\) [-] - recovery exponent (kinematic hardening).

An appropriate numerical program based on equations (1)-(8) is developed to predict material response for 1D loading and
unloading. The explicit integration procedure is applied. All derivatives in (1)-(8) are replaced by finite differences. Huang and Khan [4] proposed the following magnitudes of the material data: $D_0 = 1.08 \text{s}^{-1}$, $n = 0.87$, $m = 0.2193$, $Z_0 = 551.99 \text{MPa}$, $Z_1 = 1.026.84 \text{MPa}$. Their model does not consider both isotropic and kinematic hardening which for simple loading process (no unloading and no cycling loading) can be neglected.

3. Numerical simulation of simple tension test

Aluminum 1100 used in numerical calculations provides low strain hardening. On the basis of the Bodner-Partom constitutive equations a numerical program is developed which allows for a discrete generation of stress-strain curves for simple tension tests at various strain rates. Some numerically generated stress-strain curves for Bodner-Partom material model are presented in Fig. 1.

![Figure 1: Stress-strain curves at various strain rates - numerical simulation of tensile test for Bodner-Partom material model](image)

4. Numerical simulation of the extrusion of 1100 aluminum

In numerical computations the Simufact.Forming program is used. Experimental stress-strain characteristics based on literature data [4] is introduced as well as material data generated by procedure based on the Bodner-Partom model. The load is applied as the mass velocity - 10m/s or as the total energy absorbed. The results of simulations obtained for material data resulting from experimental investigations and data generated numerically are compared. The extrusion problem is solved as an explicit analysis in which highly nonlinear effects and inertia forces are considered. Material deformation is forced by a single blow of the hammer. A suitable design of the punch geometry (extrusion ratio $\lambda$) allowed to obtain expected magnitude of the extrusion force which is consistent with forces existing in real collisions of vehicles. The appropriate level of absorbed energy is achieved as well. The problem is solved as an axisymmetric analysis. The diameter of aluminum bar is 140 [mm], inner diameter of punch is 70 [mm], outer diameter of punch is 100 [mm]. General view of the analyzed object is shown in Fig. 2.

![Figure 2: Proposed concept of the impact energy-absorbing device](image)

5. Results of the extrusion simulations

In this chapter some results of the extrusion simulations obtained in Simufact. Forming program are presented. In Fig. 3 presented is the plot of the punch force in terms of the punch displacement. Extrusion force obtained for the Bodner-Partom material is insignificantly larger than force provided by calculations based on the experimental data.

![Figure 3: Comparison of impact energy-absorbing device real test results with the results of numerical simulations](image)

6. Conclusions and final results

The main goal of the study was to develop initial parameters of the impact energy-absorbing device which satisfy manufacturer requirements (constant force, the amount of absorbed energy etc.). Constant force requirement can be satisfied by the backward extrusion process. The main problem in the numerical calculations is the appropriate selection of the material model which assumes elastic visco-plastic properties. The properties of 1100 aluminum are taken from experimental investigations made by Huang and Khan. Results of numerical simulations show that requirement of constant force is better met in the proposed device than in the existing device (see Fig. 3). Application of the Bodner-Partom material model in calculation of aluminum 1100 extrusion at high strain rates provides a good convergence to the experimental tests. FEM analysis of the backward extrusion allows to optimize the parameters of the designed energy-absorbing device.

References


Data preprocessing for diagnostic intuitionistic statement network

Sebastian Rzydzik*
Silesian University of Technology, Institute of Fundamentals of Machinery Design, 18A Konarskiego Str., PL-44-100, Gliwice, Poland
e-mail: sebastian.rzydzik@polsl.pl

Abstract

The paper presents a calculation methodology of statements values for the purposes of diagnostic inference based on intuitionistic network statements. The main objective of the proposed method is to determine the intuitionistic statement values on the basis of values of diagnostic signals features. The presented method was applied in diagnostic process of cogeneration plant which applies Organic Rankine Cycle (ORC).

Keywords: artificial intelligence, technical diagnostics, intuitionistic network

1. Introduction

The main goal of technical diagnostics is to recognize the condition (technical state) of technical objects using some methods and resources [4,6]. The state \( s_t \) of every technical object at a given moment of time \( t \) belongs to (usually finite) set of possible states \( ST \) assigned to this object. The simplest set \( ST \) has two main states (generally class of states): \( s_t \{ \text{good; } s_t \{ \text{faulty} \}. \) The obtained information about the technical state of the object is called diagnosis.

To recognize the state of a technical object expert systems [3] are used. One of the modules of such system is an inference module. From a set of different inference methods the intuitionistic statement network can be used here [2]. This method was derived from the theory of intuitionistic fuzzy sets (IFS) [1]. In intuitionistic statement networks each node corresponds to statements, and edges describe relationships between these statements. Generally, every statement \( s \) can be written as a set:

\[
\begin{align*}
    s &= \langle c, b \rangle, \\
    &\text{where:} \\
    &c \quad \text{– the statement content,} \\
    &b \quad \text{– the statement value.}
\end{align*}
\]

The statement content can be declarative sentence, to which is attributed one of the certain values (e.g. \( c(s) \) = “the liquid level in the tank is too high”). While the statement values is represented by the ordered pair:

\[
b(s) = \langle p(s), n(s) \rangle \quad \text{for} \quad p(s), n(s) \in [0,1],
\]

where \( p(s) \) is a degree of validity (truthfulness, acceptance) and \( n(s) \) is a degree of no validity (lack of truthfulness) of statement \( s \). Within IFS it is assumed that:

\[
p(s) + n(s) \in [0,1], \tag{3}
\]

and it is not assumed that:

\[
p(s) + n(s) = 1.0. \tag{4}
\]

It is assumed that the unrecognized statements, \( p(s) \) and \( n(s) \) are equal to zero:

\[
h(s) = <0,0> \tag{5}
\]

As a degree of nondetermination statement \( s \) defines a hesitation margin:

\[
h(s) = \begin{cases} 
1 - p(s) - n(s) & \text{if } p(s) + n(s) \leq 1 \\
0 & \text{if } p(s) + n(s) > 1
\end{cases} \tag{6}
\]

where:

\[
h(s) \in [0,1]. \tag{7}
\]

Values \( p(s) \) and \( n(s) \) of statement may be determined directly or as a result of inference process using other statements, according to the accepted knowledge model.

2. Diagnostic signals

Diagnosed technical object is observed through a signal which features define interactions between the elements of this object and interactions between the object and the environment [4]. For the purposes of diagnostic activity necessary are the diagnostic signals which are carriers of diagnostic information required to determine the technical condition of the object and/or its operating conditions (Fig. 1).

![Figure 1: Acquisition of diagnostic signals](image)

In order to acquire information contained in the measuring signals it is necessary to analyse them. The result of this are values of statistical features or functional model, deriving from a single or multiple signals in the time domain, frequency domain or in the modal domain.

The values of the diagnostic signals features belong to one of two classes – the quantitative or qualitative values. In the case of quantitative values, they may be either exact or approximate. In the case of quality values may be either ordinal (e.g. “low”, “medium”, “high”) or nominal (e.g. “the type of oil”).

* Described herein are selected results of study, supported partly from the budget of Research Task No. 4 entitled “Development of integrated technologies to manufacture fuels and energy from biomass, agricultural waste and others” implemented under The National Centre for Research and Development (NCBiR) in Poland and ENERGA SA strategic program of scientific research and development entitled “Advanced technologies of generating energy”.

---


447
Finally, values of diagnostic signals features are recorded, stored and shared in the diagnostic data repository (e.g. OPC server or SQL server).

3. Description of the method

The main objective of the proposed method is to determine the intuitionistic statement values on the basis of values of diagnostic signals features.

In the case of quantitative values of diagnostic signal features is used a transition function (Fig. 2A):

\[ \{p(s) | n(s)\} = f(E(x(t)), P) \in [0,1] \]  

(8)

where:

- \( E(x(t)) \) – the single value or set of values expected signal features,
- \( P \) – the set of transition function parameters.

In the case of intuitionistic statement, whether it is set \( p(s) \) or \( n(s) \) depends on the context of statement content. The second value of the pair \( <p(s), n(s)> \) can be assumed equal to 0 or designated as a complement to 1:

\[ p(s) = 1 - n(s) \quad \text{or} \quad n(s) = 1 - p(s), \]  

(9)

but this procedure is not recommended – IFS recommends an independent setting the value of \( p(n) \) and \( n(s) \).

In the case of statement which value is uncertain and the care about preserving this uncertainty, degrees of membership of considered value are determined to the predefined classes regarded as fuzzy sets [5] (Fig. 2D).

In practice, transformation of values of diagnostic signals features to statements values \( p(s) \) and \( n(s) \) are written as procedures in the some programming language (e.g. c#, java etc.). Collections of such procedures are stored in libraries saved in knowledge base of diagnostic system.

4. Summary

The target technical object is the cogeneration plant which applies Organic Rankine Cycle (ORC). In practical solution, the presented method is used in subsystem which is part of a diagnostic system. The results obtained from this subsystem are necessary for the diagnostic reasoning process based on the intuitionistic network statement. It is also allowed to use other methods of diagnostic inference (e.g. based on beliefs networks).

References

A concept of elastic–plastic material adaptation by the thermal–FSI simulation

Daniel Sławiński¹, Janusz Badur²

¹,² Department Conversion Energy, Institute of Fluid – Flow Machinery PAS
Fiszera 14, 80-231 Gdańsk, Poland
e-mail: daniel.slawinski@imp.gda.pl¹, janusz.badur@imp.gda.pl²

Abstract

In the paper a model of elastic–plastic material adaptation undergoing thermal cyclic loads is presented. The model was verified by the uniaxial data experiment. The experiment was performed on the normal sample. After the convergence of experiment data and validation of a correct value of strain energy dissipation, for a cyclic load only, the model was used for momentum–FSI and thermal–FSI simulations. The first analysis resolved value of stresses from the cyclic strain in a standard plate. The second simulation resolved a value of thermal stresses obtained during start–up and shut–down steam turbine from a power plant. For the simulation was a tee element selected as a tee from the live steam pipeline. As a result of a material adaptation (shakedown phenomena), it was possible to develop of a new start–up curve. The new curve assumed to be shorter time of start–up steam turbine compared with the one preferred in the Standard.

Keywords: shakedown, plasticity, thermal – FSI, steam turbine, start – up, retrofit.

1. Introduction

In the paper a concept is developed of elastic – plastic material adaptation applied to a construction element. This element has been subjected to a cyclic thermal loading. Value of thermal strains resolved by the single cycle, is assumed to exceed the point of plastic material limit.

For a proper value of a plastic field and properly conducted turbine start–up it is possible to find working elements of construction within the elastic limits. Upon a given some condition it is possible to obtain elastic–plastic adaptation of material. This phenomenon is called shakedown.

The model was also verified by a uniaxial strain tension experiment. In the experiment a plate of the standard material. This phenomenon is called shakedown.

The model was also verified by a uniaxial strain tension experiment. In the experiment a plate of the standard material. This phenomenon is called shakedown.

Results of the elastic–plastic adaptation led to a new curve of a start–up steam turbine, shorted then to a recommended maintenance Standard.

2. One by the models of energy dissipation

Introducing the relationship between density of entropy $S$ and rate of entropy flux $\mathbf{h}$ a relation by balance of entropy yields [2]:

$$ p \frac{d}{dt} n = -\text{div} \; \mathbf{h} + \rho n $$

where: $n$ is the entropy production density.

The change of rate total entropy $\dot{S}$ in the body on the volume $\Omega$ in the result of the thermal flow is:

$$ \dot{S} = \frac{d}{dt} \iiiint_{\Omega} \rho s \, dV. $$

The rate of entropy change is additively decomposed into two parts. The first part describes growth in mutual interaction of the environment, the second part describes change with the system [2]:

$$ \dot{S} = \dot{S}_x + \dot{S}_w $$

where: $\dot{S}_x, \dot{S}_w$ are flux of outside entropy, coupled with the thermal flow by the surface $A$ and flux of inside entropy coupled with changes occurred in the inside body, respectively.

We suppose that the flux of entropy $\mathbf{h}$ is equal to the flux of heat divided by temperature $\mathbf{h} = q/T$. The rate of $\mathbf{n}$ is also divided by the uncomressed transformation of heat.

The change of entropy of two parts can be respectively written:

$$ \dot{S}_x = - \iint_{\partial \Omega} q n_i / T \, dA \quad \dot{S}_w = \iiiint_{\Omega} \rho n \, dV. $$

Hence, the rate of entropy evolution takes the form:

$$ \frac{d}{dt} \iint_{\Omega} \rho s \, dV = - \iint_{\partial \Omega} q n_i / T \, dA + \iiiint_{\Omega} \rho n \, dV. $$

Due to a direction of heat flow, entropy conversion from environment can be positive or negative. Thus, reversibility or irreversibility of a process depends on the entropy change. Hence, we postulate:

$$ \dot{S}_x > 0 \; \text{ - process is irreversible} $$

$$ \dot{S}_w = 0 \; \text{ - process is irreversible} $$

$$ \dot{S}_w < 0 \; \text{ - process does not exist in environment.} $$

The principle of entropy balance states that the rate of change of total entropy cannot be smaller than the flux of entropy by a surface $A$ of a body and volumetric production entropy $(\rho n / T)$ in the body. It means that the local balance of entropy is:

$$ \frac{d}{dt} \iiiint_{\Omega} \rho s \, dV = - \iint_{\partial \Omega} q n_i / T \, dA + \iiiint_{\Omega} \rho n \, dV, $$

or the local version, after using the Gauss–Ostrogradsky’s formula, came to the eq. (1). From the balance of entropy it follows that the plastic deformation and damage in material is caused irreversible by processes and dissipation of strain energy for one cycle [1]. Hence, we put that, the production of entropy is positive: $\mathbf{n} \geq 0$. 


449
3. Referential geometry

Geometry of a standard of plate applied for a uniaxial tension experiment is shown in Fig. 1.

![Referential geometry used by the experiment.](image)

Figure 1: Referential geometry used by the experiment.

Fig. 2 presents real geometry used for a tee in live steam pipeline. This geometry was used in thermal–FSI simulation of a start–up and shut–down turbo–set in a power plant.

![Geometry of a real element from live steam pipeline.](image)

Figure 2: Geometry of a real element from live steam pipeline.

4. Discussion of results

The validation model applies an elastic–plastic adaptation of a material, using data of a uniaxial tension experiment.

Fig. 3 presents data of momentum–FSI analysis, where: $F_1$, $F_2$ are a force at elastic-plastic adaptation, and a force at a destruction of element, respectively. $F_{cr}$ is a loading due to the yield limit. Points A, B, C denote of loading cycles: 3, 20 and 50 cycles, respectively.

![Plot of the results of momentum–FSI analysis of force cycles.](image)

Figure 3: Plot of the results of momentum–FSI analysis of force cycles: a) curve of plastic strain- number of loaded cycle ($\omega_{pl} = \rho nT$), b) curve of dimensionless strain energy–number of loaded cycle [2].

Fig. 4 presents plot of the curve stress – strain from the thermal–FSI simulated cycled start–up and a shutdown real element from the live steam pipeline. Only seven thermal cycles were numerically simulated – the start–up takes only 1.5 h. and shut–down takes 2.5 h.

![Plot of the plastic field used for the force cycles.](image)

Figure 4: Plot of the plastic field used for the force cycles: a) fields of plastic zone from elastic–plastic adaptation, b) fields of plastic zone after the analysed cases [2].

5. Conclusions

Elasto–plastic adaptation in the material allows to develop a new start–up curve. The new curve assumed a shorter time of start–up steam turbine compared with the one preferred in the Standards.

References


Experimental and numerical evaluation of mechanical behaviour of composite structural insulated wall panels submitted to edgewise compression

Łukasz Smakosz1, Ireneusz Kreja2
1,2 Faculty of Civil and Environmental Engineering, Gdansk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: lukasz.smakosz@wilis.pg.gda.pl1, ireneusz.kreja@wilis.pg.gda.pl2

Abstract

A composite structural insulated sandwich panel (CSIP) is a quite novel approach to the idea of sandwich structures. A series of natural-scale experimental test is required each time a change in panel’s geometry is planned and a reliable computational tool is required to precede actual laboratory testing with virtual simulations. An attempt of creating such a tool has been made with use of a commercial FEM code ABAQUS, in order to predict a behaviour of a specific kind of CSIPs with magnesium-oxide board facings and expanded polystyrene core. Results obtained from simulations taking geometrical nonlinearity as well as material nonlinearity of both core and facing materials are presented and compared with experimental data from natural-scale edgewise compression tests of wall CSIPs.

Keywords: structural insulated panels, magnesium oxide board, expanded polystyrene, experimental analysis, FEM analysis

1. Introduction

Composite structural insulated sandwich panel (CSIP) is an advanced, novel approach to the idea of sandwich structures. It is based on the main principle of combining two materials with diametrically different properties – a light-weight, soft, thick core sandwiched between two high-strength, durable and thin facings – joined together by a sufficiently strong adhesive. CSIP’s facings are made of composite materials, making a light, durable and easy to handle prefabricated element considerably stronger, immune to biological corrosion and more durable to weather conditions. Such desirable improvements make CSIPs very attractive alternative to classical SIPs with OSB facings and broaden their field of application.

However, higher facing-core elastic moduli ratio causes CSIPs to be challenging in design. Due to a large number of possible failure modes, a series of natural-scale experimental tests is required each time a change in panel’s cross-section is planned. A reliable computational tool is required to precede actual laboratory tests with appropriate numerical simulations. The Authors of the present report applied the ABAQUS software [1] to create a suitable computational model for the CSIPs with magnesium-oxide board (MgO board) facings and expanded polystyrene (EPS) core. In order to observe a behaviour and failure modes of this kind of CSIPs a series of natural- and small-scale tests was performed on a batch of samples with a specific cross-section. A comparable FEA was performed and a comparison of computational results with experiments was presented in [2, 3].

2. Material properties

The analysed panel’s cross-section is shown in Fig. 1. It consists of two 11 mm thick MgO boards, each one reinforced with a fiberglass mesh on top and bottom surfaces, a 152 mm thick EPS core layer with approximate volumetric density of 19.5 kg/m3 and a thin, continuous layer of a polyurethane adhesive joining the layers together, adding up to a total cross-section’s thickness of 174 mm.

Material properties of EPS are quite well recognized although strongly dependent on its density, while, the available characteristics of the MgO board are very limited. To determine the mechanical properties of both materials the Authors performed a series of experimental tests on small-scale samples [3].

Elastic moduli and hardening curves for EPS were obtained in a course of uniaxial compression and tension tests – its results confirm that EPS, as every other structural foam, behaves quite differently in both of those stress-states. The MgO board was tested in uniaxial edgewise compression tests and flatwise three-point bending tests and this material also shows a significant difference in average values of mechanical properties in compression and flexure. Based on these results, presented in detail in [3], the mechanical properties for FEA were established as listed in Tab. 1.

Table 1: Mechanical properties used in FEM simulations: $E$ – elastic modulus, $\sigma_y$ – yield stress

<table>
<thead>
<tr>
<th>Material</th>
<th>$E$ [MPa]</th>
<th>$\sigma_y$ [MPa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>EPS</td>
<td>6.09</td>
<td>0.09</td>
</tr>
<tr>
<td>MgO board</td>
<td>1922</td>
<td>13.5</td>
</tr>
</tbody>
</table>

Figure 1: Analysed CSIP’s cross-section – view and scheme
In addition to values presented in Tab. 1 the following values of the Poisson’s ratios were used: 0.11 for EPS and 0.18 for the MgO board.

3. Edgewise compression experimental test

Wall panel edgewise compression tests under an eccentric load were carried out on two test samples in natural scale. The dimensions of the tested panels were 2750×1000×174 mm³ and the compressive force eccentricity was \( e = d/6 \) and \( e = d/3 \), where \( d = 174 \) mm is the panel’s thickness. Due to technical limitations, the tests were performed in a horizontal position with both ends supported on hinges. In order to obtain relatively uniform distribution of stresses at the edges, the panels were inserted into steel profiles and sealed with concrete in first case and polyurethane foam in the other. The steel profiles had the height of 165 mm and the total distance between the support points was 3080 mm. The experimental set-up is shown in Fig. 2.

The applied load and the horizontal movement of the piston were recorded and presented as force-displacement relationships in Fig. 3. Both samples were damaged by a facing cracking in the contact zone between sample and steel profiles, no buckling behaviour – neither global nor local – was observed. In both cases the load eccentricity was positioned in the lower half of the cross-section and in both cases the failure occurred in the bottom facing.

4. FEM model

A commercial FEM software – ABAQUS [1] – was used to build a numerical model depicting the two tests described above. In order to capture a nonlinear behaviour of the core and the facings in a plastic range the sample was discretized with 2D solid elements.

In each case the sample was represented as a plane stress section divided into partitions corresponding to core and facing areas. A single FE mesh divided in subsections with different material properties was employed because no pre-failure deboning was observed in the compression tests and there was no need to include any additional contact interactions between the sample’s layers. The steel profiles were substituted with a rigid body planar wire sections. These parts were assembled together and a frictionless contact interaction was used between the rigid supporting parts and the sample. Pinned boundary conditions were created at a single reference point assigned to each rigid profile.

A fine mesh of 4-node plane stress elements with reduced integration and hourglass control was used. Global mesh size of 5.5 mm, resulted in 28 elements of 1:1 proportions through the core thickness. In the facing area the mesh was refined in one direction resulting in 4 elements of 1:2 proportions through the facing thickness.

Both, the MgO facings and the EPS core were modelled as elastic-plastic Drucker-Prager material. Since both, the EPS core and the MgO board show different behaviour in compression and tension two sets of properties were defined. A ‘user-defined field’ procedure was created to generate an additional field variable (FV) in such a way that it takes values from -1 (when both principal stresses are negative) to 1 (when both principal stresses are positive). Elastic properties and a hardening behaviour for compression are used for both materials when FV=-1, while the tensile properties are assigned in EPS, and flexural ones in MgO board for FV=1. For all the other values of FV the adequate material properties are interpolated between these two data sets.

The assembly was loaded in two steps – first the dead weight was taken into account and in the second step a horizontal movement of one of the supports was forced with the geometric nonlinearity algorithm active.

The comparison of numerical and experimental results is presented in Fig. 3. Two kinds of numerical results are presented: with MgO board’s \( E_{MgO}(FV=-1)=1922 \) MPa and \( E_{MgO}(FV=1)=2800 \) MPa.

![Figure 3: Comparison of force-displacement curves](image)

5. Conclusions

The comparison illustrated in Fig. 3 shows clearly that the basic numerical model has significantly undervalued stiffness and generally underestimates the strength of the actual sample. A parametric study shows that the main reason for this is the \( E \) value used to describe MgO board’s behaviour under compression. By increasing this value with a multiplier 1.5 one obtains results being much closer to the experimental ones, and since it is still in a range of values obtained from uniaxial tests it can be justified.

The behaviour of simulated sample gives satisfactory results in terms of sample’s deformed shapes and failure modes – they both match the ones observed in experiments.

References

Quantitative and qualitative researches of flow around a bridge pillar model for the Reynolds number of transitional and turbulent flow

Katarzyna Strzelecka¹, Henryk Kudela², Aleksandra Filarowska³
1,2,3 Faculty of Mechanical and Power Engineering, Wrocław University of Technology
Wybrzeże Wyspiańskiego 27, 50-370 Wrocław, Poland
e-mail: katarzyna.strzelecka@pwr.edu.pl ¹

Abstract

Experimental quantitative (local velocity measurements by LDA) and qualitative researches (visualization by dye marker) of flow around a bridge pillar model for the Reynolds number ReDw of transitional and turbulent flow were conducted. ReDw was the Reynolds number referred to the diameter of the model (cylinder), Dw = 14.69 mm.

Keywords: water tunnel, flow around a cylinder, bridge pillar, LDA, flow visualization

1. Introduction

Experimental quantitative (local velocity measurements by LDA) and qualitative researches (visualization by dye marker) of flow around a bridge pillar model for the Reynolds number ReDw of transitional and turbulent flow were conducted. ReDw was the Reynolds number referred to the diameter of the model (vertical cylinder placed in shallow water).

It is known from literature (e.g. [1]) that vortex structure above the plate can lead to the boundary layer eruption. It seems that similar phenomenon was observed behind a model of bridge pillar placed in shallow water. Previous investigations of flow past a vertical cylinder placed in shallow water were carried out for low values of hw/Dw (hw is the water depth in the test section), e.g. hw/Dw << 1 [2] or hw/Dw = 0.5; 1; 2 [3]. Chen and Jirka conducted extensive experimental researches for the case where the cylinder diameter Dc greatly exceeded the water depth. Akilli and Rockwell carried out investigations using a combination of visualization by dye marker and a PIV technique. They presented visualization results of vortex formation behind a cylinder for relatively low velocities in test section resulting in the Reynolds number (based on the water depth hw: Re = u hw/ν) from about Re_min = 500 up to Re_max = 2500.

To put more light into the processes behind a model of bridge pillar for the Reynolds number ReDw of transitional and turbulent flow authors of the paper decided to widen experiments on that subject.

2. Experimental setup

Experiments were conducted in water channel. Schematic of experimental setup working in closed circle is shown in Fig. 1. In order to clean the water in a system part or whole mass of the water can be exchanged. It is significant when carrying out visualization by dye marker researches. Water is pumped from bottom reservoir BR by one, two or three pumps P1–P3, than it flows through the filter F, rotameter (system is equipped with four rotameters R1–R4), pressure vessel PV to damp pressure fluctuation and flow regulating valve FRV to inlet collector IC. The end of the pipe supplying the inlet collector is perforated and placed inside the collector.

Figure 1: Schematic of experimental setup

Rotameters fulfil the duties of indicators. The accurate volumetric flow rate is done by a mass measurement method. For low flow rates water is directed from the bottom reservoir BR to the top reservoir TR, where the liquid level is fixed by overfall, than it flows through the rotameter R and flow regulating valve FRV to the inlet collector IC. Next water inflows to the section consisting of two packets of drinking straws where disturbances are reduced. The length of each packet is determined in conformity with [4]. Then water flows through the Witoszynski nozzle behind whose uniform velocity profile is formed. Next, the liquid inflows to test section. Next water outflows to the outlet collector OC, measuring reservoir MR placed on scales S – to determine the flow rate by mass method - and finally to bottom reservoir BR.

The average local velocity measurements were made by Laser Doppler Anemometry. Vortex formation behind a vertical cylinder in shallow water was investigated using visualization by a dye marker.

The dye injection was placed in the base of cylinder and carried out by means of an infusion pump. Schematic of the dye marker letting in is shown in Fig. 2.
Quantitative measurements were carried out for the cylinder with a diameter $D_w = 14.69 \text{ mm}$ (a bridge pillar model), placed in the water tunnel filled to a height $h_w = 65 \text{ mm}$. Test section of the tunnel has dimensions of $100 \text{ mm} \times 100 \text{ mm}$ and length $2 \text{ m}$.

Qualitative researches performed with the use of visualization by dye marker were conducted for the same values of the Reynolds number $Re_D_w$, but for two water levels ($h_w = 65 \text{ mm}, h_w = 30 \text{ mm}$) – in order to demonstrate the differences occurring in the flow with a Reynolds number (referred to the diameter $D_w$ of the model) immutability.

Figure 4 shows images observed for $Re_{D_w} = 150$ for both water tunnel depths ($h_w$). In both cases the dye is rapidly displaced from the bottom area, and directed toward the free water surface. In the case of $h_w = 30\text{ mm}$ a clear vortex structure was observed. It occupies almost the entire space between the bottom and the water level.

4. Summary

Deformation of velocity profiles were observed for each examined $Re_{D_w}$. These deformation was decreasing with the distance from the model, in cross-section $16D_w$ all profiles were already stable. Recirculation zone behind such an obstacle was observed for each $Re_w$. Qualitative research was conducted for $h_w = 65\text{ mm}$ and $30\text{ mm}$. For all tests for $h_w/D_w = 4.44$ vortex structure reached approximately a half of $h_w$, for $h_w/D_w = 2.05$ – it occupied almost the entire space from the bottom of the free water surface. Different phenomena were observed next for the same value of $Re_w$.

References

The numerical and experimental study of welding strains in the element under the load

Piotr Szewczyk 1, Maciej Szumigala 2

1 Faculty of Civil Engineering and Architecture, West Pomeranian University of Technology
   al. Piastów 50, 70-311 Szczecin, Poland
   e-mail: szewczyk@zut.edu.pl
2 Institute of Structural Engineering, Poznan University of Technology
   ul. Piotrowo 5, 60-965 Poznan, Poland
   e-mail: maciej.szumigala@put.poznan.pl

Abstract

In the paper there preliminary results are presented of numerical and experimental analysis of stress and strain level, in a loaded element during welding. The problem is how the level of structural effort depends on weakening of element during welding and the welding shrinkage. The experiment was conducted with steel flats, overlaid by means of coated electrodes. Longitudinal and transverse welds were carried out at various effort levels of steel plates. The same problem was solved numerical by using the Abaqus software. The presented subject is important for steel and steel-concrete composite structures, that are strengthening e.g. under a partial load. Strengthening structures is a field of wider research program for both authors.

Keywords: Finite Element Method, numerical modelling, welding shrinkage.

1. Introduction

Welding shrinkage is a well-known effect. There are many methods to estimate its results, depending on technology, the method and welding parameters.

The majority of available works make it possible to calculate a welding shrinkage for un-loaded elements, e.g. during prefabrication of steel structures. However, a necessity to welded elements under the load, for example during strengthening of structures. There are only a few works that estimate the behaviour of structures in such conditions. The works [2,3] simply assume that the effect of shrinkage is independent of the stress level. The paper is an attempt to define if and how the stress level of a welding element acts on the value of welding strains at the moment of maxim heating and after its total cooling.

The presented studies are part of the project, that aims at estimation of the effectiveness of strengthening a steel-concrete composite beam under the load [1].

2. The experimental models

Preliminary tests were made on steel flats of 40x10 mm cross-section and 1000 mm length. Samples of the steel flats were taken for a tensile testing. The yield stress amounts Re=260 MPa, the tensile strength Rm=375 MPa and modulus of elasticity E=206 GPa. In the Abaqus software a three-dimensional model was created of the flat bar, that took into consideration elastic and plastic features of steel with a yield point and strain hardening.

In the experiment, a load was applied by kinematic excitation, therefore the recorded effect of welding was manifested as a variation of force in the tested element.

Two types of experiments were conducted. Firstly, an overlaying welding across an element axis in its whole width (40 mm). The second was an overlaying welding along an element axis with the weld 100 mm long. In both cases rutile-coated electrodes of 3,25 mm diameter and current intensity 140 A were used.

A view of numerical and experimental models for an oblong weld was presented in the Fig.1.

![Figure 1: The models, a) numerical, b) experimental](image)

3. Measurement and results

In each case the measurement cycle proceeded as follows:

1) Insert of an initial load: 10, 30 or 50 kN.
2) Making a weld and reading the maximum decrease of force after welding (the maximal decrease of force occurs in a few seconds after welding is done).
3) Element cooling and recording the force value.

The exemplary measurement cycle for welding along an element axis under the load of 30 kN was shown in Fig. 2.
As shown in the Fig. 2, the obtained compatibility of numerical and experimental results was satisfactory. The similar compatibility was obtained for other tests. The numerical analysis was completed in full range of loads, from zero to plastic yield. Two figures below show the obtained results. Figure 3 presents graphs for welding along an axis, Fig. 4 presents graphs for welding across an element axis. The results were presented in coordinate system: force – stress in the bar before welding.

Figure 3: The results for the longitudinal weld

The solid line shows value of force adequate to the stress inserted before the test. The dashed line presents value of force in the moment after welding, in the state of its maximum decrease (about 5-10 sec. after the end of welding). The last, dotted line shows the value of force after total cooling of the bar. On the basis of presented graphs it follows, that the level of effort during welding apparently influences the value of welding strains.

In the testing case, for the longitudinal weld even at 80 MPa stress the welding shrinkage was so small, that it was not able to exceed the initial stress. The apparent decrease of force caused by welding was visible over 150 MPa. For a transverse weld from the beginning lack of shrinkage after cooling was visible. While stress was close to the yield point the force after heating and cooling became convergent. This effect is shown better in Fig. 5. The graph presents only decreases of forces, defined as the difference between the initial load and the force after maximal heating (the dashed line) and difference between the initial load and the force after cooling (the dotted line).

Figure 5: The results for the transverse weld

4. Conclusions

The presented results were obtained on the basis of one model only. Due to a large variety of materials and welding techniques applied in industry, these results should be taken only overall. However, on their basis conclusions can be drawn:

- weakening of an element during welding and a welding shrinkage strongly depend on the effort of a flat bar during welding.
- welding across the principal stress is unfavourable thus not recommended in in practice [3].
- the presented subject will be verified in the future by different parameters and welding techniques, also by strengthening of structural elements, made of steel-concrete composite beams.

References


Dynamics of rotating pendulums attached to a hub driven by a non-ideal energy source

Zofia Szmit1∗, Jerzy Warmiński2∗, Jarosław Latalski1∗
Department of Applied Mechanics, Faculty of Mechanical Engineering, Lublin University of Technology
Nadbystrzycka 36, 20-618 Lublin, Poland
e-mail: z.szmit@pollub.pl 1, j.warminski@pollub.pl 2, j.latalski@pollub.pl 3

Abstract

In the paper we consider a structure consisting of a rotating hub and two attached pendulums. The system is rotating in a horizontal plane so the gravity force does not influence its motion. Both pendulums are treated as lumped masses fixed at the ends of stiff and massless rods connected to the hub by flapping hinges. In the analysis it’s assumed that the oscillations of pendulums may be large. The considered system is driven by a DC motor with two possible output characteristics: (a) an ideal energy source where torque is given by an arbitrary defined function or (b) a non-ideal energy source, where output power of the driving DC motor is limited.

Keywords: nonlinear dynamics, non-ideal energy source, pendulums, rotating system

1. Introduction

Rotating systems are common structures in mechanical and aerospace engineering. Typical examples might be wind turbines, helicopter rotors, jet engines turbines, energy harvesters etc. Some of these systems like mechanical shredders or energy harvesters may be represented by a mathematical model comprising rotating pendulums attached to a driven hub. Even flexible blades of multiple blade rotor systems with flapping hinges at the hub may be roughly modelled by rotating pendulums as well.

Discussed above rotating systems exhibit the dependence of the system vibrations (due to the e.g. unbalance forces or blade bending) on the motion of the energy source. As reported in the literature – in case of insufficient power of the driving motor the dynamic behaviour of a non-ideal supply system may significantly differ from the corresponding idealized case [1]. This may exhibit in e.g. a strong interaction in fluctuating motor speed and large vibration amplitudes.

The discussed above Sommerfeld phenomena with respect to flexible slewing beam oscillating in longitudinal and flexural direction has been studied in the paper [2]. Different output characteristics of the driving motor was examined.

The aim of the current research is to study the system composed of two pendulums attached to the hub rotating in a horizontal plane and driven by a non-ideal energy source. Due to the vertical orientation of the hub axis the gravity force does not influence the motion of the system. Therefore, the presented problem is different from the dynamics of the classical pendulum system presented by other authors e.g. [3]. Within the frame of the presented studies the regions of possible synchronized and chaotic motions of the system are examined. Moreover, nonlinear resonance curves for the discussed system are determined.

2. Mathematical model and its equations of motion

Let us consider the structure rotating in a horizontal plane and comprising two pendulums attached to a rigid hub. Each of the pendulums is composed of a lumped mass fixed to a massless and infinitely stiff rod that is connected to the hub by a flapping hinge joint represented by a nonlinear Duffing spring and a viscous damper (Fig. 1). The radius and the mass moment of inertia of the hub are denoted by \( R_0 \) and \( J_0 \) respectively; \( m_j, l_j \) are mass and length of the \( j \)-th pendulum, \( c_j \) is a viscous damping coefficient and \( k_1, k_2, k_3 \) are the coefficients of a nonlinear Duffing type spring, where \( j = 1, 2 \) (see Fig. 1). The hub may rotate or oscillate in horizontal plane and its current position is described by an angle \( \psi \). The torque driving the system under consideration is generated by the DC motor, which we consider in two variants: (a) as an ideal energy source where the output torque is defined by a time dependent function or (b) as limited power supply source, where the actual motor characteristic is taken into account. Each pendulum may oscillate around the hinge and its relative motion is described by a coordinate \( \varphi_j \) defined with respect to floating frame of reference rotating with the hub. We note that constraints in both hinges allow a full rotation of the pendulums in their relative motion.

![Figure 1: Model of the structure rotating in a horizontal plane, view from the top](image)

The equations of motion of the system are derived from Lagrange equations of the second kind. The resulting expressions are transformed to their non-dimensional form by introducing dimensionless time \( \tau = \omega_0^0 t \), where \( t \) is time and \( \omega_0^0 = \sqrt{\frac{c}{m \omega_0}} \) is a natural, linear frequency of the first pendulum. Finally, the three equations of motion are:

---

∗The support of the Structural Founds in the Operational Programme Innovative Economy (IE OP) financed from the European Regional Development Fund research project "Modern material technologies in aerospace industry” number POIG.01.01.02-00-015/08 is gratefully acknowledged.
where \( j = 1, 2 \) and:

\[
\begin{align*}
\delta_{0j} &= \frac{R_0}{J_j}, \\
\delta_j &= \frac{R_1}{J_j} = \sqrt{\frac{\delta_{0j}^2}{\delta_j} + 1 + 2\delta_{0j} \cos \varphi_j}, \\
\gamma_{0j} &= \frac{m J_j^2}{J_j}, \\
\gamma_j &= \gamma_0 \frac{d_j}{d_j}, \\
\nu_j &= \frac{\kappa_j}{J_j}, \\
\kappa_j &= \frac{c_j}{J_{0j}}, \\
\mu &= \frac{m_j J_j^2 \omega_{0j}^4}{J_{0j}^3}.
\end{align*}
\]

The derived system of Eqs (1) and (2) is a general one and allows the comprehensive studies on dynamic behaviour of the structure, including nonlinear phenomena effects due to large rotations and arbitrary driving torque function.

3. Numerical and analytical studies

Derived dimensionless differential equations of motion are strongly nonlinear. Therefore, dynamics of the system was studied by direct numerical simulations. At first possible chaotic motions of the system were examined. This has been found for the ideal energy source case (\( \mu = \rho \cos \Omega \tau \)) where \( \rho \) is excitation amplitude – see Fig. 2(a) and for the limited power supply case as well – Fig. 2(b). In this second case the driving torque was defined by \( \mu = \rho \cos \Omega \tau - E_2 \psi \) formula where \( E_2 \) comes from DC motor characteristic. The given bifurcation diagrams of \( \varphi_j \) against excitation amplitude \( \rho \) indicate that the chaotic motion of the system (the dark zones) is observed for different excitation amplitudes. This observation was confirmed by Poincaré maps – see [4].

Figure 2: Bifurcation diagram of \( \varphi_j \) against excitation amplitude \( \rho \) and fixed frequency \( \Omega = 1 \), zoom of the first chaotic region (a) ideal energy source, (b) non-ideal energy source.

Next, the resonance curves were plotted. This analysis has been performed for an ideal energy system with both perfectly tuned pendulums. Selected resonance curves obtained for different amplitudes of excitation \( \rho \) are given in Fig. 3(a), where dashed lines represent unstable regions. For higher excitation amplitudes sub-harmonic resonances were observed.

These numerical solutions were verified by harmonic balance method. For lower excitation amplitudes the analytical results fully coincide with numerical ones. However, for the case of \( \rho = 0.3 \) new analytical solutions were found. Therefore, these new outcomes have been checked numerically by setting different initial conditions. Results of this analysis are shown in Fig. 3(b).

The later numerical analysis showed that close to the top of the resonance curve the solution is unstable – see Fig 3(b). Furthermore, additional stable solutions, not found previously, are present. These are located on the inner loop of the curve. Therefore, for certain excitation frequencies e.g. \( \Omega = 1.4 \) even seven different solutions may exist.

4. Conclusions

The derived differential equations of motion of the system are strongly nonlinear. The analysed bifurcation scenario showed harmonic oscillations, then period doubling and finally the zone of chaotic motion for the ideal and non-ideal energy source as well. However, for the non-ideal power supply system the zone of chaotic motion is much smaller.

Plotted resonance curves have shown sub-harmonic resonances and unstable zones when the amplitude of excitation is increased. The given outcomes of this research follow the initial studies of the considered system presented in the former authors paper [4].

References


Numerical analysis of resistance of the aluminium and concrete composite beam with shear connectors created as solid elements

Szumigala Maciej¹, Łukasz Polus²

¹,² Faculty of Civil and Environmental Engineering, Poznań University of Technology
Piotrowo 5, 60-963 Poznan, Poland

e-mail: szumigala.maciej@put.poznan.pl¹, lukasz.polus@put.poznan.pl²

Abstract

In the article numerical analysis of the resistance of the aluminium and concrete composite beam with shear connectors created as solid elements is presented. Attention is focused on the modelling of the shear connector. An aluminium and concrete beam consists of a concrete slab, steel sheeting, connectors and an aluminium beam. The Abaqus program was used to simulate a four point bending test of a composite beam. As a result of the analysis, the resistance of the composite aluminium and concrete beam was obtained and compared with the plastic resistance moment of the composite cross-section obtained using EN 1994-1-1.

Keywords: aluminium and concrete composite beam, shear connector

1. Introduction

Modern designers look for new solutions serving as parts of a sustainable construction [1]. Aluminium and concrete structures are composite structures which are not so widespread as steel and concrete composite structures [5]. Mromliński, Mazzolani and Mandara worked on this issue in the previous century [3][4]. However, aluminium and concrete structures were not commonly used because of the lack of a connector which would effectively join an aluminium beam and a concrete slab. This problem was solved by a special connector [6]. Yet other reason for a low popularity of these structures is still unsolved. Aluminium alloys are 5 to 6 times more expensive than steel [2]. Aluminium and concrete structures may be used in buildings where the risk of corrosion is high, for example in swimming pools.

2. Problem formulation

Numerical analysis of the resistance of the aluminium and concrete composite beam was presented in [5]. However, shear connectors were created as beams there. In the article, shear connectors are created as solid elements and presented in Figure 1.

The model consists of an aluminium beam, shear connectors and a concrete slab on a profiled steel sheeting (see Figure 2). A concrete slab was created with eight-node cuboidal finite solid elements, an aluminium beam was created with four-node shell elements. Steel sheeting serves as a skin for the concrete slab.

Figure 1: An aluminium beam with the shear connectors

The model consists of an aluminium beam, shear connectors and a concrete slab on a profiled steel sheeting (see Figure 2). A concrete slab was created with eight-node cuboidal finite solid elements, an aluminium beam was created with four-node shell elements. Steel sheeting serves as a skin for the concrete slab.

Figure 2: An aluminium and concrete composite beam

Physical laws for each material are shown in figures below.

Figure 3: Stress-strain relations for materials

Calculations were performed using the Abaqus program and the Newton-Raphson method. Load was applied in the form of displacement like in a four-point bending test. It was assumed that the resistance of composite beam is reached when there is a local extreme on the static equilibrium path. In order to verify the resistance moment of the composite beam obtained in a numerical analysis the plastic resistance moment of the composite cross-section was determined using [7].
3. Results

The result of the analysis was a strain energy curve with a local extreme. The strain energy curve is presented in Fig. 4, the curve of the force at one of the points of displacements application is presented in Fig. 5.

![The strain energy curve](image1)

**Figure 4: The strain energy curve**

![The curve of the force at one of the points of displacement application](image2)

**Figure 5: The curve of the force at one of the points of displacement application**

The stresses in the aluminium beam are presented in Figure 6 and the stresses in the concrete slab are presented in Figure 7.

![A map of the equivalent Huber-Mises-Hencky stresses](image3)

**Figure 6: A map of the equivalent Huber-Mises-Hencky stresses**

The stresses in the aluminium beam were greater than the yield strength of the aluminium.

![A map of the main stresses S33](image4)

**Figure 7: A map of the main stresses S33**

Neutral axis of the composite beam is above the steel sheeting in the concrete slab. There are locally high stresses in place of connection between a concrete slab and rigid plates used to apply loads. Physical laws are connected with equivalent stresses for this reason main stresses may be higher than the assumed values.

The resistance of the composite aluminium and concrete beam obtained in numerical calculations is 72.9kNm. The plastic resistance moment of the composite cross-section using Eurocodes is 64.8kNm.

4. Conclusions

The conducted analysis showed that it is possible to prepare a numerical model of an aluminium and concrete composite beam with an accurate model of the shear connector. The resistance of the composite aluminium and concrete beam obtained in numerical calculations is similar to the plastic resistance moment of the composite cross-section obtained by Eurocodes.

References


Numerical analysis of the beam to column end – plate bolted connection

Maciej Szumigala¹, Katarzyna Ciesielczyk²

¹,² Faculty of Civil and Environmental Engineering, Poznań University of Technology
Piotrowo 5, 60-965 Poznań, Poland

E-mail: maciej.szumigala@put.poznan.pl¹, katarzyna.ciesielczyk@put.poznan.pl²

Abstract

A new method of computations of the end – plated connection was presented in the standard EN-1993-1-8. It is definitely more complicated and occupying than the method presented in the standard used previously in Poland (PN-B-03200: 1990). According to the rules given in Eurocode 3 it is required to determine the moment – rotation characteristic in order to determine the following properties: moment resistance, rotation stiffness and rotation capacity. The main aim of the paper is the development of finite element models of end – plate connection. Different numerical models were presented in order to find the best model corresponding with the analytical results. The analytical procedure was performed according to the rules described in the European standard.

Keywords: beam to column end – plate connections, Eurocode 3, finite element method, structural joints

1. Introduction

According to the conventional method joints can be divided into idealized two groups [1]:
- fully rigid with complete rotation capacity,
- pinned transferring no bending moments.

However, in the real engineering constructions no ideal joints occur. A joint can be more rigid or more pinned but it is always between this two extreme solutions. Wherefore, in the designing code [2] it is required to determine the moment – rotation characteristic in order to determine the moment resistance, rotation stiffness and rotation capacity which allows the optimal designing of the joint.

Finding this properties is also important because of the economics reasons. The beam – column bolted connections are widely used in steel structures. The cost of the connection accounts for a major part of the costs of the whole structure. Wherefore, it is really important to design them optimally as it is possible.

Analysis performed in the paper was divided into two parts. The first step consists of the analytical computations according to the rules presented in the Eurocode 3 part 1-8 [2]. The assumption of the second step was to find a numerical model sufficiently corresponding with the results of the analysis and will present the moment – rotation characteristic as well as possible.

2. The geometry of the model

The beam column joint with a bolted end – plate connection is the subject of the analysis. The joint consists of two hot rolling I-beams: IPE 300 (beam), HEB 160 (column) and steel plate welded to the end of the beam section. Dimensions of the joint were presented in the Fig. 1. The elements are made of steel with yield strength of 355 MPa and ultimate tensile strength of 490 MPa.

These elements were connected by 6 bolts with diameter of 20 mm spaced in 3 rows. The 8.8 class of the bolts was used.

Figure 1: The dimensions of the joint

The analysis was performed under a static load. The load was applied at the end of the beam. In order to find the bearing capacity in the numerical model the load was applied as a vertical displacement.

3. Numerical modelling

The prepared models can be ordered from the most simplified model to the most detailed model. The authors found a simplified model corresponding with the real M - φ characteristics of joint.

The analysis of different types of numerical modelling were performed. The steel elements (I-beams and steel plate) were modelled with shell and solid elements. The fasteners are modelled in very different ways, e.g. [3, 4]: beam elements, connector elements, solid elements, rigid elements and spring elements. Up till now, the authors focused on the first three methods of modelling. Moreover, according to the Abaqus Lecture [5] in the finite element modelling in Abaqus the bolt can be replaced by fasteners interaction using a beam type of connection.
3.1. The simplified models

In the first group of models the elements of a joint were modelled with shell finite elements. In this group the bolts were not modelled. The elements were joined in the points where the axes of bolts should be by:
- beam elements as wire parts,
- connector elements – beam type of connection,
- fasteners,
- tie coupling.

In Fig. 3 the moment – rotation characteristic of some of the analysed simplified models was presented.

4. Conclusion

As presented shortly, there are many ways exist to model the beam to column end plate bolted connection. The bolts can be modelled with different detailing, and they can be connected with the I-beams in very different ways. Using each of these models the similar (but different) results can be obtained.

In the next step all proposed models will be compared in order to find the most optimal numerical model.

References


Reliable mechanical characterization of layered pavement structures

Andrea Venier¹, Tomasz Garbowskii²*

¹,²Faculty of Civil and Environmental Engineering, Politecnico di Milano
Piazza Leonardo da Vinci 32, 20-133 Milano, Italy
e-mail: andrea.venier@polimi.it, tomasz.garbowski@polimi.it

²Institute of Structural Engineering, Poznań University of Technology
Piastrowa 5, 60-965 Poznań, Poland
e-mail: tomasz.garbowski@put.poznan.pl

Abstract

In order to obtain geometric and mechanical properties of an axisymmetric layered system a numerical high performance model was constructed built on the data provided by the falling-weight-deflectometer (FWD). This apparatus allows for a fast non-destructive testing and consists of a falling mass hitting the pavement and a set of sensors collecting the vertical surface oscillations at different distances. The objective of this research is at first to construct a forward model for simulating wave propagation into a layered media. Adopting the spectral element technique only one element per layer is required and computational efficiency is guaranteed, especially when compared to standard dynamic FEM procedures. The main numerical efforts at this stage are the attainment of the dynamic complex stiffness matrix for each discrete frequency and wave number and the solution of the related linear system. Subsequently an inverse method based on the framework of the forward model is proposed for the calculation of the unknown parameters. The procedure is now concerned with the minimization of the objective function which quantifies the difference between computed quantities and their measured counterpart. The solution of the nonlinear system of equations is investigated by iterative methods and finally results of different algorithms are compared.

Keywords: falling weight deflectometer, spectral element method, inverse analysis

1. Introduction

Pavement consists of a few layers of asphalt, placed on granular sub-base and base, all of them specified by their thickness and elastic properties. However the initial properties are gradually changing during an extensive overloading of the road structure, which enforces frequent in-situ examinations of the deteriorated pavement characteristics. This is mainly done using nondestructive tests enhanced by the numerical or analytical models and inverse analysis.

Mechanical identification of layered pavement structures is an actual and an important problem which attracts many researchers around the world. In the literature exist many approaches based on an inverse procedure combined with a dynamic test (e.g. falling-weight-deflectometer - FWD) and analytical or numerical models. In most cases static analysis is used, creating an obvious divergence between a purely dynamic response of the real structure and its numerical quasi-static model. This problem was partially solved introducing the filtered (i.e., zero-frequency) force and deflection values [2] into the inverse procedure. Other solutions usually base on dynamic FE models, which unfortunately are very costly and therefore impractical in real life applications.

2. Problem formulation

The proposed here procedure uses a spectral element method (SEM) instead of dynamic FE models. The main advantage, besides a substantial decrease of the computational time, is a significant increase of experimental data to be incorporated into the inverse analysis. This is mainly to freely sample the dynamic response not only in space but also in time. Moreover the bigger amount of data creates a great possibility to regularize a usually ill-posed inverse procedure.

In the following subsections the forward model based on SEM is briefly described followed by the short introduction to inverse analysis and concluding remarks.

2.1. Wave equations

A vertical impulse load acting on a homogeneous isotropic half space generates axisymmetric perturbations. Therefore adopting a cylindrical reference system it is possible to combine Navier’s equations and the Helmholtz potential decomposition in order to obtain wave motion equations. Denoting the potentials (ϕ and ψ) and the vertical and radial coordinates (z and r), the governing differential equation reads:

\[
\frac{\partial^2 \phi}{\partial r^2} + \frac{1}{r} \frac{\partial \phi}{\partial r} + \frac{\partial^2 \phi}{\partial z^2} = \frac{1}{c_p^2} \frac{\partial^2 \phi}{\partial t^2} \tag{1}
\]

\[
\frac{\partial^2 \psi}{\partial r^2} + \frac{1}{r} \frac{\partial \psi}{\partial r} + \frac{\partial^2 \psi}{\partial z^2} = \frac{1}{c_s^2} \frac{\partial^2 \psi}{\partial t^2} \tag{2}
\]

where:

\[
c_p = \left(\frac{\lambda + 2\mu}{\rho}\right)^{1/2}, \quad c_s = \left(\frac{\mu}{\rho}\right)^{1/2} \tag{3}
\]

while the displacement field respectively in radial and vertical downward direction is:

\[
u = \frac{\partial \phi}{\partial r} - \frac{\partial \psi}{\partial z}, \quad w = \frac{\partial \phi}{\partial z} + \frac{1}{r} \frac{\partial (r \psi)}{\partial r} \tag{4}
\]

The way to solve the differential equations (1-2) is Fourier transform. Shifting from time domain to frequency reduces the problem to Bessel equations: its solution is the Bessel’s function J⁰
In order to discretize the domain it is necessary to impose zero amplitude at distance \( r = R \) far enough from the origin.

\[
J_0(kR) = 0 \tag{5}
\]

Choosing the constants (wavenumbers) as \( k_m = \alpha_m / R \), where \( \alpha_m \) represents the \( m \)-th root of \( J_0 \), the solution of equation (5) is achieved. The potentials in the frequency domain are consequently defined as:

\[
\varphi_{mn}(r, z) = A_{mn} e^{-ik_{p,m}z} J_0(k_m r) \tag{6}
\]

\[
\psi_{mn}(r, z) = B_{mn} e^{-ik_{s,m}z} J_1(k_m r) \tag{7}
\]

where:

\[
k_{p,m} = \left( \frac{\omega^2}{c^2} - k_m^2 \right)^{1/2}, \quad k_{s,m} = \left( \frac{\omega^2}{c_s^2} - k_m^2 \right)^{1/2} \tag{8}
\]

being \( k_{p,m} \) the wavenumber in vertical direction, \( k_{s,m} \) the wavenumber in radial direction, \( A_{mn} \) and \( B_{mn} \) constants defined by boundary conditions, \( J_1 \) the Bessel’s function of the first kind.

### 2.2. Discrete spectral solution

Thanks to the linearity and homogeneity of the governing equations it is possible to use the principle of superposition. This means that by double summation over \( M \) wavenumbers and \( N \) angular frequencies \( \omega \), one can reconstruct the whole vibration system that vanishes at \( r = R \).

\[
\varphi_{mn}(r, z, t) = \sum_{m} A_{mn} e^{-ik_{p,m}z} \sum_n J_0(k_m r) e^{-i\omega_n t} \tag{9}
\]

\[
\psi_{mn}(r, z, t) = \sum_{m} B_{mn} e^{-ik_{s,m}z} \sum_n J_1(k_m r) e^{-i\omega_n t} \tag{10}
\]

Summation over \( N \) can be done by means of FFT. At this point it is straightforward to build the stiffness matrix for the layer elements from the nodal displacement obtained by (4) and the applied boundary tractions. The process is required for every wavenumber and every frequency while the global stiffness matrix is assembled in the same way as in the finite element method. The final result is the following:

\[
u(r, z, t) = \sum_{m} \hat{u}_{mn}(z, k_m, \omega_n) \hat{F}_0 J_0(k_m r) \hat{F}_n e^{i\omega_n t}
\]

\[
w(r, z, t) = \sum_{m} \hat{w}_{mn}(z, k_m, \omega_n) \hat{F}_0 J_1(k_m r) \hat{F}_n e^{i\omega_n t}
\]

being \( \hat{u}_{mn} \) and \( \hat{w}_{mn} \) the displacements for a unit load condition, while \( \hat{F}_m \) and \( \hat{F}_n \) represent the Fourier-Bessel spatial coefficients and fast Fourier time coefficients of the load.

### 2.3. Inverse analysis

In order to bring the details of the proposed procedure one needs to discuss also the general framework of inverse analysis and minimization algorithm. Herein the brief explanation of main features of the inverse procedure followed by a detailed elucidation of implemented minimization algorithms is presented.

Back-calculation analysis with a particular application to constitutive model calibration is a tool widely used by many researchers (see e.g. [3, 4]). In general it merges the numerically computed \( \hat{U}_{\text{num}} \) and experimentally determined \( \hat{U}_{\text{exp}} \) measurable quantities for a discrepancy minimization. A vector of residual \( \mathbf{R} \) in time \( t \) can be constructed in the following way:

\[
\mathbf{R}(\mathbf{x}) = \hat{U}_{\text{exp}} - \hat{U}_{\text{num}}(\mathbf{x}). \tag{11}
\]

This measures the differences between the aforementioned measurable quantities. By adjusting the constitutive parameters (encapsulated in the vector \( \mathbf{x} \)) embedded in the numerical model, which in turn mimic the experimental setup, an iterative convergence towards the required solution can be achieved. The minimization of the objective function \( \omega \) (within the least square frame) takes the form:

\[
\omega^2 = \sum_{i=1}^{n} \left( R_i^2 \right)^2 = \left\| \mathbf{R} \right\|^2, \tag{12}
\]

and is usually updated through the use of first-order (gradient-based) or zero-order (gradient-less) algorithms. Among many first-order procedures based on either the Gauss-Newton or the steepest descent direction in a nonlinear least square methods, the Trust Region Algorithm (TRA) seems the most effective. The TRA uses a simple idea, similar to that in Levenberg-Marquardt (LM) algorithm (see e.g. [7]), which performs each new step in a direction combining the Gauss-Newton and steepest descent directions.

Here another great tool is selected for an automatic update of the model parameters prediction, which is based on Bayes principles, namely Kalman filter. By making use of such algorithm one achieves not only the parameter estimates but also their uncertainty.

### 3. Concluding remarks

Herein a procedure based on the discrete spectral solution has been implemented and its verification has been obtained by comparing the results with those obtained in [3,4]. Special consideration went into performance since code efficiency is crucial for inverse analysis algorithms. Therefore static correction technique, interpolation method of FRF and high performing libraries (LAPACK) were used. Calculations were conducted over an Intel Pentium T2330 1.60 GHz and 2 GB of RAM memory and code has proved to be efficient enough: about 0.12s for each forward computation, which is few hundred times faster then FE forward model. Additionally adopting the Kalman filter the overall identification procedure gives a realistic perspective for fast and robust engineering application.

### References


Complex modal analysis for time-variant dynamical problems of rotating pipe conveying fluid

Lihua Wang¹, Zheng Zhong²*
¹,² School of Aerospace Engineering and Applied Mechanics, Tongji University
Shanghai, 200092, P.R.China
e-mail: lhwang@tongji.edu.cn, zhongk@tongji.edu.cn

Abstract

Conventional complex modal analysis can only deal with time-invariant problems. Numerical methods relying on the finite difference method in time domain cannot ensure the time continuity in computation for time-variant problems. Therefore, a new scheme of complex modal analysis for the time-variant dynamical problems of rotating pipe conveying fluid system is presented in this paper. The appropriate orthogonality conditions are constructed to decouple the time-variant equation of motion. Complex frequencies and modes of vibration are analytically formulated. The complex mode superposition method is used for the dynamic analysis in the time and frequency domains. The changes in the fundamental frequencies and damping are numerically evaluated. Numerical time-variant examples of rotating pipe conveying fluid illustrate the effectiveness and accuracy of the proposed method. The proposed solution scheme is also applicable for any other time-variant dynamical problems.

Keywords: complex modal analysis, time variant, dynamical problem, rotating pipe conveying fluid

1. Introduction

Pipe conveying fluid system is a typical time-variant system which is widely used in many mechanical engineering applications, such as water supply and sewage, petroleum transmission etc. Its vibration characteristics and dynamic stability are of critical importance. Many numerical techniques are investigated to study the dynamics of pipe conveying fluid system, such as Galerkin method, multiple scales method, power series method etc. For the rotating flexible pipe conveying fluid model, the solution techniques are mainly depending on the Galerkin method and Runge-kutta method. For time-variant problems, the damping is time-variant and usually non-classical, and so the natural frequencies and modes of vibration are complex and the equations of motion have to be decoupled in the complex domain. The technique of complex modal analysis was first developed by Lee [1]. After that it has been introduced for the complex analysis of the rotating systems and simply supported beams.

Modal analysis method is very popular for solving time-invariant linear equations of motion. However, for a time-variant dynamical problem, use of modal analysis in its well-known form has never been investigated. In this paper, we present a complex model analysis for the time-variant rotating pipe conveying fluid system. An analytical procedure for the complex frequencies and modes of vibration are developed and the orthogonality conditions to decouple the equation of motion are derived. Analytical responses of damped linear vibrating systems to arbitrary excitation are obtained.

2. Discretization for the dynamical problem of rotating pipe conveying fluid

Consider a rotating flexible pipe conveying fluid model. The governing equation of motion derived in [2] is

\[
\left( M_1 + M_2 \right) \ddot{i} + 2 M_2 \dot{\omega} - \left( M_1 + M_2 \right) \omega \dot{i} + \left( M_1 \ddot{\theta} + M_2 \dot{\theta} \right) \omega ' = \left( M_1 \ddot{i} + M_2 \dot{i} \right) \omega
\]

(1)

where \( M_1 \) is the mass per unit length of the pipe, \( M_2 \) is the mass per unit length of the fluid, \( u \) is the transverse displacement, \( \omega \) the angular speed of the pipe rotation, \( L \) is the length of the pipe, \( EI \) the bending stiffness of the pipe and \( t \) the time. Based on the classical separation of variables technique, the solution of Eq. (1) can be written as

\[
u(x,t) = \sum_{j=1}^{n} d_j(t) \varphi_j(x) = \sum_{j=1}^{n} d_j(t) \varphi_j(x)
\]

(2)

where \( n \) is the number of included modes, and \( d_j(t) \) are the non-dimensional temporal functions in generalized coordinates, \( \varphi_j(x) \) can be expressed as

\[
\varphi_j(x) = \sin \theta x - \sinh \theta x + \sin \theta \delta L + \sinh \theta \delta L (\cosh \theta x - \cos \theta x)
\]

(3)

where \( \theta \) can be obtained from \( 1 + \cos \theta \delta L \cosh \theta L = 0 \). Substituting (2) into governing equation of motion (1), multiplying it by \( \varphi_j(x) \), and integrating the equation from \( x = 0 \) to \( L \), after that the discretized equation of motion for the system can be formulated as

\[
M(t) \ddot{d}(t) + C(t) \dot{d}(t) + K(t) d(t) = F(t)
\]

(4)

The initial conditions for Eq. (4) are given as

\[
d_0 = \int_0^L \varphi_j(x) y(x,0) dx, \quad \dot{d}_0 = \int_0^L \varphi_j(x) \dot{y}(x,0) \varphi_j(x) dx
\]

(5)

and \( y(x,0) = \gamma_0 \varphi_j(x) \); \( \dot{y}(x,0) = \gamma_0 \varphi_j(x) \).

*This work is supported by National Natural Science Foundation of China (Project No. 11202150), Fundamental Research Funds for the Central Universities and Shanghai Leading Academic Discipline Project (Project No. B302).
3. Complex modal analysis

Consider a general dynamical problem with time-variant coefficients of the form described in Eq. (4), where \( M(t) \) is the mass matrix, \( C(t) \) is the damping matrix and \( K(t) \) is the stiffness matrix. \( F(t) \) is the arbitrary excitation matrix. When \( M(t) \) is a non-singular matrix, Eq. (4) can be reformulated as

\[
d = -M^{-1}Cd - M^{-1}Kd + M^{-1}F
\]

Then we define \( y = \begin{bmatrix} d \end{bmatrix} \).

Combining (4) and (7) renders

\[
y = Sy + f(t)
\]

where

\[
S = \begin{bmatrix} -M^{-1}C & -M^{-1}K & I \\
1 & 0 & 0
\end{bmatrix}, \quad f(t) = \begin{bmatrix} M^{-1}F \\
0 \\
0
\end{bmatrix}
\]

The initial conditions are

\[
d(0) = d_0, \quad d(0) = d_0.
\]

From (7) and (10) we can obtain

\[
y_y = y(0) = \begin{bmatrix} d_0 \\
d_0
\end{bmatrix}
\]

With the initial conditions (11), solution for (8) can be expressed as

\[
y(t) = e^t Y_y + e^t \int_0^t e^{-t'} f(t') dt'
\]

where \( P = \begin{bmatrix} 0 & S \end{bmatrix} \). Define \( \xi \) as the roots of the following eigenfunction

\[
[P - \xi I]Y_y = 0
\]

where the eigenvectors corresponding to eigenvalues \( \xi \) are \( y_y \), i.e.

\[
(P - \xi I)Y_y = 0
\]

and \( y_y \) satisfy the following condition

\[
y_y^T Y_y = 1
\]

Define the generalized mode matrix as

\[
Y = \begin{bmatrix} y_1, y_2, \ldots, y_n \end{bmatrix}
\]

Combining Eq. (14) and (16) we obtain

\[
PY = Ydiag(\xi_1, \xi_2, \ldots, \xi_n)
\]

Since the eigenvalues are diverse, the eigenvectors are linear independent. Eq. (17) can be rewritten as

\[
Y^T PY = diag(\xi_1^2, \xi_2^2, \ldots, \xi_n^2)
\]

Eq. (18) can be rewritten as

\[
Y^T e^t Y = e^{tY} = e^{diag(\xi_1, \xi_2, \ldots, \xi_n)} = diag(e^{\xi_1}, e^{\xi_2}, \ldots, e^{\xi_n})
\]

thus

\[
e^t Y = Ydiag(e^{\xi_1}, e^{\xi_2}, \ldots, e^{\xi_n}) Y^T
\]

Substituting (20) into (12) renders the response of a system

\[
y(t) = Ydiag(e^{\xi_1}, e^{\xi_2}, \ldots, e^{\xi_n}) Y^T y_y + \int_0^t Ydiag(e^{\xi_1}, e^{\xi_2}, \ldots, e^{\xi_n}) Y^T f(t') dt'
\]

4. Numerical examples

For a rotating pipe conveying fluid model, the same material coefficients are used for solution as in reference [1] for comparison and validation. The material coefficients are: \( E=7.8x10^3 \text{ Pa}, I=1.21x10^{-7} \text{ m}^4, L=0.025 \text{ m}, M_1=1.88x10^{-3} \text{ kg/m}, M_2=1.6485x10^3 \text{ kg/m}, \omega_0=50 \text{ rad/s}. \) The initial conditions are \( u_0=0.001 \text{ m}, \dot{u}_0=0 \text{ m/s}, \) and the velocity can be expressed as

\[
v(t) = v_0 \left( e^{v_0' t} + e^{-v_0' t} \right)/2 \text{ or } v(t) = -\alpha \omega \sin \omega t - v_l \cos \omega t.
\]

Figure 1: Numerical solution for the tip displacement and eigenvalues of the system when \( v_0 = 0.02 \text{ m/s} \)

Fig. 1(a) shows a numerical solution for tip displacement of the pipe when \( v_0=0.02 \text{ m/s}, \) which demonstrates that the numerical solutions obtained from complex modal analysis are in good agreement with the solutions from Runge-Kutta method. Fig. 1(b) illustrates the 1st-4th eigenvalues of the dynamical system. The real parts of the eigenvalues describe the damping of the system and the imaginary parts characterize the frequencies of the system. When \( v_0=0.02 \text{ m/s}, \) the frequencies are increasing, and the absolute value of the negative damping is increasing which results in the decrease of the amplitude of the system vibration. The numerical solution for tip displacement of the pipe when \( v_0=0.02 \text{ m/s} \) are presented in Fig. 2(a), which verifies the effectiveness and accuracy of complex modal analysis once again. Fig. 2(b) displays the eigenvalues of the dynamical system, which reveals the damping of the system is first increasing when \( v>0 \) and then decreasing when \( v<0 \), while the frequencies is always increasing during the vibration.

5. Conclusions

Complex modal analysis associated with assumed modes is introduced for the time-variant dynamical problem of rotating pipe conveying fluid. Numerical solutions are compared with the solutions obtained from conventional Runge-Kutta method which demonstrates that the proposed method can achieve good accuracy for solving time-variant dynamical problems. Complex eigenvalues obtained in the complex modal analysis can clearly describe the variation tendency of the damping (or negative damping) and frequencies of the system. The proposed method provides a general solution scheme for time-variant dynamical problems.

References


Circular membrane as a voltage signal generator

Aleksandra Waszczuk-Młyńska¹, Stanisław Radkowski²

¹,² Faculty of Automotive and Construction Machinery Engineering, Warsaw University of Technology
Narbutta 84, 02-531 Warsaw, Poland
E-mail: awm@mechatronika.net.pl¹, ras@simr.pw.edu.pl²

Abstract

In this work, the experiment was shown, applying a vibrating circular membrane/diaphragm to generate a voltage signal. On the basis of the obtained results a correlation filter was created, the Hilbert transforms were applied using time signals.

Keywords: signal analysis, Hilbert transform, signal correlation, laser vibrometer

1. Introduction

There are numerous methods of signal analysis, including, e.g. various kinds of transforms, correlations. All these transformation methods are intended to obtain the greatest possible amount of information about the time signal, e.g. how the signal behaves, what are the signal characteristic frequencies. These methods can also help in diagnostics, detecting possible faults [2,3].

2. Experiment

The subject of this research was a circular membrane with a piezoelectric material glued onto it in the middle Fig. 1.

Figure 1: Circular membrane

The membrane was activated to vibrate by an acoustic wave from a speaker. The activation was white noise with the frequency range from 0 to 500 Hz. The system (the membrane) response was tested with the help of a laser vibrometer, which uses a Doppler effect in its measurements. Also by means of this device, the visualisation of the results was obtained. Figure 2 shows the frequency spectrum and the first form of vibrations.

Figure 2: Frequency spectrum and the first form of vibrations

In the subsequent stage of the experiment, a programme was created in the LabView, for the analysis of the piezoelectric material response, the voltage change, and the spectrum structure, obtained as a result of vibrations of the membrane activated with white noise. Figure 3 illustrates visually of the programme results.

Figure 3: Visualisations of the programme results
3. Analysis of the results

In the previously described experiment, a signal of the membrane response is taken, which means its vibration response to extorsion and the piezoelectric material responses, i.e. of the voltage obtained as a result of the membrane vibrations. The figure below shows the spectra, the blue colour denotes the piezoelectric material spectrum, the red colour indicates the spectrum of the membrane response.

Figure 4: Signal spectrum

3.1. Correlation filter

The correlation of two signals \( f(x) \) and \( g(x) \) is a function

\[
\varphi_{fg}(x) = \int_{-\infty}^{\infty} f(x)g^*(x-x) \, dx
\]

where \( g^* \) is a conjugate function.

The correlation filter is a correlation of two signals, where on the obtained data Fourier transform was used, which corresponding to the product. This filter allows for a signal reinforcement for the overlapping frequencies.

Figure 5 shows the use of the correlation filter on these signals.

Figure 5: Correlation filter on the signals obtained from the membrane and piezoelectric material vibrations

3.2. Hilbert transform

The Hilbert transform was shown on the obtained signals[1].

\[
f(t) = \frac{1}{\pi} \int_{-\infty}^{\infty} \frac{f(x)}{x-t} \, dx = f(t) \ast h(t)
\]

(2)

\[
h(t) = \frac{1}{\pi t}
\]

(3)

\[
SA = f(t) + jH[f(t)]
\]

(4)

Figure 6 illustrates the characteristic results of the tested signal, resulting from the Hilbert transform.

Figure 6: (a) Hilbert transform envelope, (b) Spectrum from the Hilbert transform envelope, (c) Dependence of the real part of the Hilbert transform from the imaginary part

4. Summary

Applying the correlation filter on time signals of the membrane responses and on the piezoelectric material provides the reinforcement of the characteristic frequencies. The Hilbert transform and its properties allow for the variation of signal rendering.

References


Development of a crashworthiness standard for assessment of cutaway buses

Jerzy W. Wekezer¹, Michal Gleba², Jeff Siervogel³*

¹,²,³ Department of Civil and Environmental Engineering, Florida A&M University – Florida State University
College of Engineering, 2525 Pottsramer Street, Tallahassee, FL 32310-6046, USA

e-mail: jwekezer@fsu.edu¹, mggleba2@fsu.edu², jsiervogel@fsu.edu³

Abstract

Research efforts in responding to needs of cutaway bus industry and paratransit bus operators are described in the paper. These efforts utilize non-linear, dynamic computational mechanics, finite element methods, and nonlinear, explicit dynamic computer code LS-DYNA. Computational mechanics tasks were carried on to support design, fabrication and use of custom made equipment for experimental testing, and to support verification and validation of the finite element (FE) models. The paper shows how computational mechanics was used to develop a mature standard for crashworthiness and safety assessment of cutaway buses.

Keywords: crashworthiness assessment, rollover, cutaway bus, FEM, computational dynamics, LS-DYNA, experimental testing

1. Introduction

Population of the state of Florida approaches 20 million residents making it the fourth largest in the USA. It is a popular home state for many retirees, who make 18.7% of the total number of the state residents, with over 70,000 of them living in nursing homes. Large number of the elderly (65+) cannot drive their own cars because of their age and medical conditions, and they rely on local paratransit bus transportation. These buses are also referred to as cutaway buses due to their unique fabrication process. They are becoming increasingly popular; Public Transit Office of the Florida Department of Transportation (FDOT) alone acquires over 300 such buses annually. In addition, the cutaway buses are also purchased by nursing homes, schools, churches, airports, car rental companies and other organizations. An example of a small cutaway bus is shown in Fig.1, left.

1.1. Unique features of cutaway buses

The cutaway buses are mid-size vehicles carrying usually from ten to twenty four passengers. They are frequently prepared to accommodate two disabled passengers on their wheelchairs. They are made in two stages. In the first, chassis with the driver cab and a frame are made by reputable bus companies per current safety standards. Then, in the second stage, they are modified by smaller businesses which specialize in outfitting them with custom-made passenger compartments. These companies, referred to as “body builders”, are often too small to afford supporting their own research and development (R&D) units, as the large automakers do. Therefore, their design differ from one manufacturer to another in terms of consistency in passenger protection and safety.

1.2. Safety standards

Dimensions of the cutaway buses and their two-stage manufacturing process made them exempted from safety standards which were developed for smaller passenger cars as well as for large coaches. To fill this gap, the “body builders” try to demonstrate the strength of their bus roof structures by using FMVSS 220 standard, [3,5], which follows conservative quasi-static load tests for school buses in the US. However, more advanced, dynamic based safety standards (Regulation 66, [5,11]) were developed in Europe. They are based on dynamic rollover testing which more closely resemble actual rollover accidents. This standard was endorsed by 44 countries (without US) through the United Nation resolution [11]. Its concept was also adopted by the Florida Standard described in this paper.

2. Finite element analysis of cutaway buses

Concerns about passenger safety of cutaway buses used in Florida have stimulated its FDOT Transit Office to support research efforts to identify the best and safest buses, and to set up high industry standards. As a result, such research has been carried out at the Crashworthiness and Impact Analysis Laboratory (CIAL) at the Florida A&M University – Florida State University joint College of Engineering over the past fifteen years.

2.1. FE models developed

Extensive computational mechanics studies were carried out at CIAL, thus far for six different cutaway buses [2,5,7,8,9,10]. The developed models exemplify a wide range of vehicles purchased, and represent 83% of manufacturers of buses acquired in 2010. Models developed cover buses with wheelbase ranging from 3505 mm to 6477 mm. The smallest FE model is depicted in Fig.1, right.

All of the FE models of the buses were developed in two separate stages [1]. First, a FE model of the cutaway chassis was acquired from public domain (either a Ford Econoline Van, or a Ford F250 model), developed by the National Crash Analysis Center (NCAC) at George Washington University [4]. The FE model was revised to match the specification of the chassis used for the given bus.

Next, the bus cage model was developed using specifications obtained from the manufacturer. In the past, bus manufacturers used a 2D drafting software, such as AutoCAD, which resulted in a lengthy process of developing and meshing 3D models of bus cages [1]. In recent years, many cutaway bus manufacturers have changed their design process and transitioned into a 3D modeling software such as Autodesk Inventor. This has substantially improved the speed of the finite

*The project presented in this paper has been sponsored by the Transit Office of the Florida Department of Transportation through a series of multi-year contracts beginning in 1999.
element model development, since the geometry development process was reduced. Inventor files, after a short modification and cleanup, were transferred to the Hypermesh package for the FE mesh development.

At the final stage, the assembly process was carried out using LS-PrePost software, which is a default pre- and post-processor for LS-DYNA package. This effort resulted in a complete cage body structure. Finally, the FE model of the body was attached to the modified model of the chassis.

All of the developed FE models of paratransit buses have been equipped with water ballast dummies to simulate passenger load during rollover. They are shown in yellow (Fig.1, right) and are seat belted.

2.2. Verification and validation

All FE models of the buses were verified and validated (V&V, [1]) to make sure that conclusions obtained are solid and reliable. It required to develop an extensive testing program at different, multi-scale levels, [7,10]. The program was developed based on computational mechanics analysis.

3. Florida Standard

Based on several years of research the first Florida Standard for crashworthiness assessment of cutaway buses was approved by FDOT and was implemented in 2007. Since then, the Transit, Research, Inspection and Procurement Services (TRIPS) program of the FDOT Transit Office uses it in support of decision making procedure in its cutaway bus acquisition process. The most current Florida Standard was further simplified based or recent data obtained from computational mechanics analysis and experimental testing. In order to sell cutaway buses, a bus manufacturer must secure a pre-qualification status (Pre-Qual). It requires a selected bus to pass the following five steps: 1) drawing review, 2) frame evaluation, 3) wall-to-floor test, 4) wall-to-roof test, and 5) panel test. Upon successful completion of all steps above, the bus maker is allowed to sell its product within the first year of the contract during which it has to carry out and pass a full-scale rollover test.

4. Conclusions

The program presented in this paper benefits several constituencies:

a. FDOT receives a research-based advice regarding possible acquisition of the best buses.
b. Transit passengers in Florida and beyond travel in much safer buses with smaller chances for casualties and injuries during possible accidents.
c. Bus manufacturers receive much needed guidance and advice on how to improve their product.
d. Sponsored research supports graduate students pursuing M.S. and Ph.D. degrees in dynamic behaviour of steel structures and systems.
e. CIAL disseminates results to research community through high quality journal publications and conference proceedings.

References

Consistency and reliability of the surface texture measurement results obtained with different measuring methods

Marta Wiśniewska*
Faculty of Mechatronics, Warsaw University of Technology
Św. A. Boboli 8, 02-325 Warsaw, Poland
e-mail: martwisn@mchtr.pw.edu.pl

Abstract

Determination of surface geometrical features is crucial for the evaluation of tribological properties and wear of industrial elements. In effect, high accuracy and reliability of surface texture measurements has to be ensured. In order to meet this demand, numerous methods of surface geometry investigation have been devised. In spite of the fact that a growing interest in non-contact methods, such as a white-light interferometry, is observed, the profilometry (a stylus method) remains the most popular one. However, there is still nearly no information concerning reliability and repeatability of the results given with the use of methods mentioned above. There is also insufficient data concerning the consistency of the surface texture parameters obtained with profilometers and white-light interferometers, as the researches referring to this issue were limited to a certain type of industrial, non-standard surfaces. Therefore, the author found it necessary to compare performance of the instruments of the kind. The calibration artefacts recommended by ISO 25178 series were used as the reference surfaces. The results of the research, indicating on some difficulties with ensuring reproducibility of the surface texture measurements, are presented in the paper.

Keywords: white-light interferometer, stylus profilometer, surface texture, reliability, reproducibility, consistency

1. Introduction

The surface roughness of industrial components has an enormous impact on their performance, i.e. their tribological properties, wear and, in result, operational reliability or cost of service. Due to the rapid development of nanotechnologies, the importance of surface texture investigation become fundamental.

Neither scientists nor manufacturers of advanced measuring equipment could remained indifferent to this necessity for assessing surface texture properties with the accuracy higher than ever before. As a result, numerous methods of surface measurements were devised. One of the most interesting is a white-light interferometry which gives an opportunity to measure surface roughness with an exceptional resolution – both vertical and horizontal one - that is combined with an extensive measuring range exceeding a few millimetres.

A stylus method of surface roughness examination, which remains the most popular one, was also improved. An introduction of interferometric measuring gauges [5] ensuring an extra-ordinary range to resolution ratio of the profilometers allowed to conduct measurement of curved or steep sloping surfaces.

However, in spite of these remarkable developments, there is nearly no information concerning the repeatability and reliability of the measurement results obtained with the use of methods mentioned above. To make things worse, the researches referring to consistency of the surface parameters values given with white-light interferometers and profilometers were limited to measurements of specific, industrial and non-standard surfaces. Thus, an influence of non-homogeneity of a surface geometry could not be separated from an impact of the measuring method itself. Therefore, the author found it necessary to compare credibility and repeatability of the surface texture parameters measurement with a use of these measuring techniques.

The instruments used in the research were the CCI SunStar white-light interferometer [1] and the Form Talysurf PGI 830 profilometer [3], both by Taylor Hobson – a leading manufacturer of advanced surface measuring instruments.

2. Materials and methods

In order to minimise an influence of the measured surface non-homogeneity on the research results, two types of calibration standards recommended by ISO 25178 series [4] were used as reference surfaces. In spite of numerous artefacts proposed in this set of normative documents, only groove, rectangular (PGR) and periodic triangular shape (PPT) material measures are applicable to be used to assess characteristics of both profilometers and white-light interferometers. In order to assess repeatability of the results acquired with use of these instruments, each standard was measured ten times.

2.1. Groove, rectangular (PGR) surfaces

These reference surfaces have a wide groove with flat bottom that is characterised with its nominal depth $d_{\text{nom}}$ [4]. In the experiment, two standards of this type, differing in their nominal depth, were used. Their characteristics are presented in Table 1. According to the standards’ calibration certificates the uncertainty of $d_{\text{nom}}$ estimation equals to 5% (level of confidence 95%).

<table>
<thead>
<tr>
<th>Manufacturer</th>
<th>Material</th>
<th>$d_{\text{nom}}$ [µm]</th>
<th>$d_{\text{nom}}$ [µm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Taylor Hobson</td>
<td>Glass</td>
<td>2.55</td>
<td>9.40</td>
</tr>
<tr>
<td>Taylor Hobson</td>
<td>Glass</td>
<td>2.55</td>
<td>9.40</td>
</tr>
</tbody>
</table>

Table 1: Metrological properties of PGR reference surfaces

2.2. Periodic triangular shape (PPT) surfaces

These material measures reproduce triangular shape and they should be assessed with the use of $Ra$ [4], $Rz$ [2] and $Rt$ [2] parameters. In the research, two standards of the kind were applied. Their properties are outlined in Table 2. According to the standards’ calibration certificates the uncertainty of aforementioned roughness parameters estimation equals to 5% (level of confidence 95%).

*This work was supported by Polish Society of Theoretical and Applied Mechanics and the statutory funds of Faculty of Mechatronics WUT.
Table 2: Metrological properties of PPT reference surfaces

<table>
<thead>
<tr>
<th></th>
<th>PPT-1</th>
<th>PPT-2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Manufacturer</td>
<td>Mahr</td>
<td>Mahr</td>
</tr>
<tr>
<td>$Ra$ [$\mu$m]</td>
<td>3.10</td>
<td>2.39</td>
</tr>
<tr>
<td>$Rz$ [$\mu$m]</td>
<td>10.47</td>
<td>9.47</td>
</tr>
<tr>
<td>$Rt$ [$\mu$m]</td>
<td>10.52</td>
<td>9.56</td>
</tr>
<tr>
<td>Material</td>
<td>Glass</td>
<td>Steel</td>
</tr>
</tbody>
</table>

## 3. Results

In the chapter, the results are presented in box-whiskers charts.

### 3.1. Groove, rectangular (PGR) surfaces

Sample measurement results obtained in the research, when PGR-2.55 standard was used, are presented in Fig. 1. The thin solid and dotted lines correspond with nominal depth $d_{nom}$ values and the uncertainty intervals referring to them, as stated in Subsection 2.1.

Figure 1: Measurement results obtained for PGR-2.55 surface

Then, as the variances of $d$ values obtained with both instruments are not equal, a Kruskal-Wallis test was performed in order to evaluate if the differences between the given results are the ones of statistical importance.

This test proved that an impact of the measurement method is statistically important for both groove surfaces ($p<<0.05$). On the other hand, all results are within the uncertainty limits of nominal depth estimation. Therefore, there is no reason for assuming that one measurement method is more reliable than another one.

### 3.2. Periodic triangular shape (PPT) surfaces

The measurement results given in the research, when PPT-2 standard was used, are presented in Fig. 2 and Fig. 3, similarly to the values obtained for PGR specimens.

![Figure 2](Image)

Figure 2: $Ra$ parameter values obtained for PPT-2 specimen

As the charts indicate, the results obtained with a profilometer and a white-light interferometer are inconsistent.

Also, only the roughness parameter values given with stylus method are within the range limited by the uncertainties of nominal parameters’ estimation. $Ra$ values measured with CCI SunStar are underestimated, whereas both $Rz$ and $Rt$ values were overestimated, when optical measuring method was used.

The repeatability of the measurement results received with Form Talysurf PGI 830 is significantly better than the one of results obtained with the white-light interferometer.

## 4. Conclusions

All the inconsistencies mentioned above clearly outline how demanding task it is to ensure reliability of the surface texture measurement results. In consequence, it is essential to indicate the method of surface geometrical properties assessment to be applied. It is worth indicating in technical product specification which method of surface topography measurement should be used in order to assess surface properties. Without applying these solutions, the correct interpretation of the results and, in effect, predicting wear and reliability of the engineering surfaces of the cooperating mechanical objects is extremely difficult or even impossible.

## References


A redundantly actuated 4RRR planar parallel manipulator and sensitivity of its trajectory inexactness to inertia parameters of its limb

Wiktoria Wojnicz¹, Krzysztof Lipiński²
¹,² Faculty of Mechanical Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: wwojnicz@pg.gda.pl¹, klipins@pg.gda.pl²

Abstract

In the paper a summary of dynamics research is presented for a redundantly actuated 4RRR planar parallel manipulator. This search is focused on numerical Multibody modelling used to model dynamics of the manipulator. In order to obtain the required motion of the manipulator, a model-based controller is introduced. Two numerical models are necessary to work simultaneously. The first refers to the observed object. In the second one (included in the controller), additional constraint equations are present. The requested motion of the platform is described by use of these constraints. However, the number of the driven joint exceeds the number of the system degrees of freedom. To obtain the requested torques, the Moor-Penrose pseudo-inverse of a non-square matrix is implemented in the controller. As slightly inaccurate inertia parameters are introduced in the controller model for one of the limbs (the limb is a modifiable subsystem attached to the main construction), resulting trajectories differ from the requested ones. To correct the values of inertia parameters a single test of a circular trajectory is proposed.

Keywords: multibody modelling, redundantly actuated manipulators, identification, parallel manipulators, planar manipulators

1. Introduction

In the research, dynamics of a planar multibody system is investigated. Tests are restricted to numerical calculations only. Two aspects are essential: redundant actuation of mechanical systems (a), imprecise knowledge of values of some inertia parameters (b). The first aspect is complex, but the solutions are known from the scientific literature. In most cases, to predict the driving torques, an inverse dynamic model is introduced in the model-based predictive controller. Unfortunately, contrary to the traditional methods of torques estimation, the inverse of a square matrix can not be used in the present case (corresponding matrices are non-square for the redundantly actuated systems). In the literature, the most frequent solution is to operate with the right Moore-Penrose pseudo-inverse. However, a proper and accurate numerical model of the observed system is essential. When the real parameters of the observed object differ from these used in the controller, the real trajectories are realised inaccurately. The additional trajectory closed-loop controller has to work intensively, or the obtained trajectories can differ significantly from the requested ones. Alternatively, inertia identification procedure can help to predict the torques correctly. The present goal is to find sensibility of the trajectory errors to the known errors of the controller model.

2. Multibody model of the considered manipulator

For a dynamic modelling, the 4RRR planar manipulator is considered a multibody structure [3, 4] composed of 20 rigid bodies (Fig. 2). Its reference body $\emptyset$ is motionless. Massless one-degree-of-freedom translational and rotational joints are used only (all multi-dof connections are modelled by chains composed of joints and massless bodies). Additional constraints are added to the system to joint ends of its kinematical chains. The joint displacements are taken as system generalized coordinates, $\mathbf{q}$. Next, the Newton/Euler’s equations of dynamics are developed for free body diagrams, and combined with the kinematic equations of kinematical chains. The obtained vectors are projected on joint axes and the obtained matrix form of the system dynamics is [1-4]:

$$\mathbf{M}(\mathbf{q}) \cdot \dot{\mathbf{q}} + \mathbf{F}(\mathbf{q}, \dot{\mathbf{q}}) - \mathbf{Q}(\mathbf{q}, \mathbf{q}^T, \mathbf{t}^r, \mathbf{t}, \mathbf{t}^e) = 0,$$  

where: $\mathbf{M}$ – mass matrix; $\mathbf{F}$ – column matrix of inertia components depend on velocity squares; $\mathbf{Q}$ – column matrix of generalized forces based on external forces, $\mathbf{F}$, and torques, $\mathbf{t}^r$, and time, $\mathbf{t}$.

![Figure 1: Details of the considered 4RRR parallel manipulator: sketch of its structure, limbs numbering and trajectory (a); numbering of the system bodies (b).](image)

As constrains are present in the system, a loop cutting procedure is introduced, and dynamic equations of the reference tree structure are developed and extended with constraint interactions. Moreover, related algebraic constraint equations, $\mathbf{h}$, [1-4] are introduced. It leads to [1-4]:

$$\mathbf{M}(\mathbf{q}, \dot{\mathbf{q}}) \cdot \ddot{\mathbf{q}} + \mathbf{F}(\mathbf{q}, \dot{\mathbf{q}}) + \mathbf{Q}(\mathbf{q}, \mathbf{q}^T, \dot{\mathbf{t}}^r, \dot{\mathbf{t}}, \dot{\mathbf{t}}^e) + \mathbf{J}^T(\mathbf{q}) \lambda = 0,$$  

where: $\mathbf{J}$ –s the Jacobian matrix of constrains $\mathbf{h}$; $\mathbf{A} = \mathbf{J} \cdot \dot{\mathbf{q}}$.

Equations (2) constitute a base for two numerical models. The first is used to model the observed object. The second is used to simulate the controller (is implemented in the model-based controller).
Additional set of the constraint equations is necessary (user-constraints) in the controller. They describe the requested motion of the platform \([2-4]\):

\[
\mathbf{h} = \mathbf{0}; \quad \mathbf{J} \cdot \mathbf{q} - \mathbf{v} = \mathbf{0}; \quad \mathbf{J} \cdot \dot{\mathbf{q}} + \mathbf{A}^* \left( \mathbf{F}_u - \mathbf{J} \mathbf{q} \right) = \mathbf{0}.
\]

where: 
- \(\mathbf{v}\) is a column matrix set of the platform’s velocities;  
- \(\mathbf{a}\) is a column matrix set of the platform’s accelerations;

According to Eqn (3), kinematics of the system is described uniquely. All the joint positions, velocities and accelerations are computed. These kinematic relations are used in a dynamics model implemented in the controller. However, the number of the driven joint exceeds the number of the system degrees of freedom. Solution of the problem is not unique, i.e., an infinite set of combinations can effect in identical accelerations of the system. At the present considerations, the joint driving torques are obtained from

\[
\mathbf{Q}_x = \mathbf{A}^* \left( -\mathbf{M}_a + \mathbf{J}_s^T \mathbf{J}_s \mathbf{M}_a \right) \mathbf{q} + \mathbf{A}^* \left( \mathbf{F}_u - \mathbf{J}_s^T \mathbf{F}_s \right),
\]

where: 
- \(\mathbf{A}^* = (\mathbf{A} \cdot \mathbf{A}^T)^{-1}\) is the Moor-Penrose pseudo-inverse of a non-square matrix \(\mathbf{A}\), \([1,2,3,5]\);  
- \(\mathbf{q}\) – rows of the matrix that correspond to the dependent coordinates;  
- \(\mathbf{a}\) – its rows that correspond to the independent coordinates.

3. Tests conditions and results of numerical analysis

A sketch of the considered manipulator is presented in Fig. 1a-b. Contrary to the classic case (actuations installed in the limb/reference joints) limb/reference and limb/platform joints are actuated. Three of the limbs \((L1, L2\) and \(L3\) in Fig. 1a) are actuated. Limb \(L4\) is passive (driven). Inertia parameters of the limb \(L4\) are unknown (the limb is a modifiable subsystem attached to the main construction). Its geometrical parameters are measurable easily, but precise identification of its inertia parameters is impossible. It is considered that constant proportion is observed between the limb \(L4\) inertia parameters and a single inertia factor is sufficient to evaluate all the inertia parameters of the limb \(L4\). The geometrical and inertia parameters of the rest of the system are considered as known (obtained in a set of accurate tests done during the assembling).

In the numerical model introduced in the controller, an initial estimation of the inertia is proposed and kept unchanged during the performed numerical tests. Contrary, in the model of an observed object, the related inertia parameters are considered variable, i.e., they differ between different tests. The obtained trajectories differ from the required (dynamics of the model of the observed object and the non-accurately estimated torques).

The behaviour due to different values of the inertia factor are tested numerically. A circular trajectory is requested from the platform centre. The obtained trajectories (Fig. 3) differ from the circular one (Fig. 3a, 3c). The inaccurate inertia factor may not be the only source of the trajectory error. Additional changes can be associated with elasticity components present in the joints. Such additional elasticity component is introduced at joint #20. It results in trajectory errors visualised in Fig. 3b. The character of these changes differs from the previous. Trajectory errors collect at different instants of time differ too. It gives a hope to identify values of the unknown parameters simultaneously (i.e., in a single test done at the initial phase of the trajectory) to improve behaviour of the model-based control algorithm, when the imprecisely known parameters are replaced by the identified values in next instants of the simulations.

4. Conclusions

The presented results confirm sensitivity of the trajectory errors to the nature of model error. In order to improve the motion, additional closed-loop controller should be installed. A circular trial motion is an alternative. It can help to identify the model errors and to improve the control. Error sensitivity is confirmed in the paper. Future investigations will be focused on the methods of potential identification.

References

Deformation analysis of the Kościuszko Mound in Cracow

Bogumił Wrana¹, Natalia Pietrzak²
¹,² Faculty of Civil Engineering, Cracow University of Technology
Warszawska 24, 31-155 Kraków, Poland
e-mail: bwrana@interia.pl, nati@silbud.pl

Abstract

In the paper the slope stability problem of the Kościuszko Mound in Cracow, Poland is considered. The slope of the cone is not uniform in all directions, on the surface of the cone are pedestrian paths. The soil parameters were adopted in accordance with the detailed geological soil testing performed in 2012. Calculating model includes geogrids. The upper part was covered by MacMat geogrid, while the lower part of the Mound was reinforced using Terramesh Matt geogrid. The slope analysis was performed by successive reduction of $\phi/c$ parameters.

Keywords: computational models of soil mechanics, influence of irrigation on static mound deformation

1. Introduction

The Kościuszko Mound located in Cracow has a complex external surface geometry with irregular, soil layers as well as the groundwater head. The figures below illustrate this difficulty. Starting from the bottom of the mound the soil layers are introduced to computational model. Mechanical and physical parameters of the layers were included in the calculation in accordance with the Geology Engineering Report consisting of drilling and cone penetrating testing results [4].

Figure 1: Finite element mesh

Figure 2: Soil layers

Generally the mound is made of cohesive soils with limestone layers at the bottom. The Mohr-Coulomb and Cam-Clay macromechanical models of plasticity were used for soil layers. The two main model parameters appearing in the yield criterion and strain hardening/softening rule are the friction angle $\phi'$ and cohesion $c'$ [1,2]. The Hoek-Brown rock model [3] as a non-linear approximation of the strength of rocks was introduced to the limestone base layers of the mound. The material behaviour of rock may be different from the numerical model, generally it may be stiffer and stronger. Furthermore, rock may also show a significant tensile strength.

2. Simplification of computational model

Layers of soil described in the previous section have a complex geometry. Such a number of layers of complex shape leads to element mesh excess computing capabilities. Two steps were taken in order to simplify the real model of the mound. Soil layers with similar geotechnical parameters have been merged and their contact layers tailored according to the Plaxis capabilities of computing.

3. Finite Element Method calculations

The slope of the mound cone is not uniform in all directions and there are pedestrian paths on the surface of the cone. The area near pedestrian paths required finer mesh while the base area of the mound (limestone) did not require this. The mesh generation process by itself also takes into account the soil stratigraphy as well as all structural objects, load or boundary conditions. The 10-node tetrahedral elements of the 3D mesh of Plaxis program are used.

The inner part of the mound drainage as well as the outer surface of the mound stabilized with geogrids were also modelled as in Figure 3 and 4. The upper part was covered by MacMat geogrid, while the lower part of the mound was reinforced using Terramesh Matt geogrid.

As a step of calculation the rainfall during the flood of 2010 was modelled. A total of 33 days of precipitation is presented in the chart below and applied in Plaxis application.

4. Finite Element Method results

In the slope analysis the strength parameters $\tan \phi$ and $c$ of the soil are successively reduced until slip failure of the structure occurs. The dilatancy angle $\psi$ is, in principle, not affected by the $\phi/c$ reduction procedure. However, the dilatancy angle can never be larger than the friction angle. The strength of...
structural objects like geotextile reinforcement and anchors, if used, are not influenced by $\phi/c$ reduction.

Figure 3: The mound inner drainage

Figure 4: Two types of geogrids on the surface

Figure 5: Rainfall definition

The total multiplayer $\sum M_{sf}$ definition is used to define the value of the soil strength approximation at a given stage in the analysis:

$$\sum M_{sf} = \frac{\tan \phi_{input}}{\tan \phi_{reduced}} = \frac{c_{input}}{c_{reduced}}$$  \hspace{1cm} (1)

The calculation results show that the differences in the results of stability before and after irrigation are very small (see Fig. 6) and are greater in the case of irrigated mound. Tendency of a structure to tilt to the East in both cases can be observed.

Figure 6. Safety factor result for both cases- before and after heavy rainfall

5. Conclusion

The authors’ intention was to estimate the stress and strain state by using the actual computational methods and estimate characteristic value of slope stability factor. Figure 6 shows changes of deformation with decrease of soil parameters, and one can observe that:

- for both cases, before and after heavy rainfall the results are similar,
- shape of curvatures indicated that the total multiplier $\sum M_{sf}$ is proportional to reduce the cohesion $c$ and tangent of the friction angle $\phi$,
- it can be observed, that very small part of curves describe elastic range (app. 1%), the others describe constant unlimited hardening of soil layers,
- the constant growing deformation after having reached the limit value of $\sum M_{sf}$ is not observed, that means that for this ground structure it is not possible to obtain characteristic value of the safety factor based on $\sum M_{sf}$.

6. References

Diagnostics of historic columns using wave propagation

Monika Zielińska¹*, Magdalena Rucka²*

¹ Faculty of Architecture, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: monika.zielinska@pg.gda.pl

² Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
e-mail: magdalena.rucka@pg.gda.pl

Abstract

This paper presents a numerical analysis of elastic wave propagation in columns of historical buildings for diagnostics purposes. Numerical calculations were performed using the finite element method in the Abaqus software package. The analysis was carried out for three types of brick columns: a full column, a column filled with debris, and a column empty inside. The excitation was in the form of a wave packet and signals of propagating waves were registered at selected points. The influence of the internal structure of columns on registered signals and wave propagation maps was studied.

Keywords: diagnostics, historic columns, elastic wave propagation, numerical analysis, finite element method

1. Introduction

The columns, as very slender elements, transfer a large load in relation to their dimensions. Especially in Gothic, the columns were very high, what allowed the facility to create of structures characterized by an open space character. In the past, many cases were known in which a lack of the pillar stability led to the collapse of either a part, or even the entire object. The instances of Saint Mary’s Church in Gdańsk or Saint John’s the Baptist Church in Gdańsk may serve as an example where the collapse of one of the pillars caused a significant damage to the vaults. The catastrophic failures of columns are mostly caused by the material degradation under the influence of time but also by moisture or by applying additional loads. In such circumstances, pillars require repair and strengthening. However, documentation of historic buildings often is lacking in material and structural data, therefore non-destructive diagnostic methods are of great importance in revitalization process of historic buildings.

The paper discusses the elastic wave propagation method in diagnostics of historic columns. The diagnosis of pillars can allow for the assessment their condition, the recognition of materials of which they are made, the identification of the internal structure and the identification of the occurrence of moisture. Additionally, it can serve as a basis for the pillars reconstruction, modernization or maintenance project.

2. Diagnostics using wave propagation method

Currently, diagnostic methods based on wave propagation enjoy a high interest. Those methods are especially popular in the case of historic buildings due to their non-destructive nature. However, the obtained results are usually hard to interpret, thus in situ measurements are often preceded by the performance of numerical models based on the finite element method. Such studies were conducted, inter alia, on the columns in the Mallorca cathedral [3], Cathedra of Noto [1] and in the temple of S. Nicolò l’Arena [2] and they allowed for recognition of the internal structure of the columns.

3. Wave propagation in columns

In this paper, numerical models of the columns were performed in the Abaqus software package. The calculations were carried out for the three types of brick columns: the full column (Fig. 1a), the column filled with debris (Fig. 1b), and the column empty inside (Fig. 1c). The pillars had a shape of an octagon, with a 60 cm side length. All models were meshed using S4R shell elements. The number of nodes was equal to 90582, 91663, 58308 and the number of elements was equal to 91498, 92019, 58394, for the full, debris-filled, and empty inside column respectively. As an excitation, a 5-cycle wave packet with the central frequency of 80 kHz was used. The force \( p(t) \) was applied at the edge of the column and the acceleration signal of propagating waves was registered at point 1, towards the \( y \) axis (Fig. 2).

Figure 1: Geometry of considered columns: a) full brick column, b) brick column filled with debris, c) empty inside brick column

*Calculations were carried out at the Academic Computer Center in Gdańsk.
4. Results

The wave propagation signals in time domain calculated at point 1 were presented in Fig. 3 for the full column, the column filled with debris, and the empty inside column, respectively. In addition, the acceleration maps at two selected time instances ($t = 0.3$ ms and $t = 0.7$ ms) were illustrated in Fig. 4, showing wave propagation in the selected cross-section of columns. The disturbance spreads radially from the point where the force was applied. Consequently, the waves reflect from the surfaces that limit the column. The reflection occurs at the free edges of the column, at column breakings, and at the surface of the internal air cavity and debris contact. The course of acceleration signals is therefore an effect directly caused by the wave propagation, as well as the reflection from the column surfaces.

5. Summary

The performed analyses are the first stage of works aiming at non-destructive diagnostics of historic columns using wave propagation method. The influence of the internal structure of columns on registered signals and wave propagation maps was studied. Further works will be directed to experimental measurements of wave propagation in mortar columns.

References


Experimental and numerical analysis of wave propagation in ground anchors

Beata Zima¹, Magdalena Rucka*²

¹² Faculty of Civil and Environmental Engineering, Gdańsk University of Technology

Narutowicza 11/12, 80-233 Gdańsk, Poland

e-mail: bezima@pg.gda.pl¹, mrucka@pg.gda.pl²

Abstract

The study presents the results of experimental and numerical research of elastic wave propagation in steel rods embedded in concrete, which can be considered models of ground anchors. The main aim of this research is the non-destructive diagnostics and the assessment of the state of ground anchors, using the guided wave propagation method. Laboratory models of anchors with different bonding lengths were tested and voltage signals of propagating waves were registered at several locations. For all tested specimens corresponding numerical modeling were created. Results of calculations were compared with experimental signals. Characteristic amplitude changes caused by wave reflections interesting from the point of view of diagnostic process were identified and indicated.

Keywords: non-destructive diagnostics, ground anchors, elastic wave propagation, experimental investigations

1. Introduction

Guided waves are stress waves, which follow a path defined by the boundaries of the structure [6]. The efficiency of the application of guided wave propagation in non-destructive testing methods has been proved in many previous pieces of research [2,5,7]. Wave propagation has also a great potential in diagnostics of these structural elements, which state cannot be assessed on the basis of standard visual inspection, for example ground anchors. A main role of ground anchors is supporting the stability of excavations by transferring tensile forces into surrounding ground layers with high load capacity. The ground anchor consists of a steel tendon, which can be made of a rod, a multi-wire strand or a cable and an anchor body, which is formed in the subsoil by injecting grouting mortar [4].

Despite very restrictive acceptance tests and high strength of reinforced concrete elements, the real problem of ground anchors is their durability. They are subjected to continuous deterioration of their state. The most common problem is creeping of soil or steel, which results in stress release. The ways to avoid the danger are reducing the load supported by the anchorage [3] or monitoring the level of stresses in the tendon. Another real threat is corrosion of the tendon because of a constant contact with ground water. Poor ground conditions or poor workmanship can also lead to deterioration of bond between the anchor body and the tendon. In contrast to the problem of stress release, the process of corrosion or bond deterioration cannot be easily monitored and controlled.

Acceptance tests allow the evaluation the object directly after its realization and they are not carried out regularly. Hence, there is a need to develop diagnostic methods dedicated to constant monitoring of such type of structures. Non-destructive diagnostic techniques based on elastic wave propagation allowing for the assessment of the state of elements, objects or parts of structures, which are not easy available have been recently the subject of a growing interest [1,8,9].

The study deals with the numerical and experimental research of guided wave propagation in laboratory models of ground anchors. The anchors with different lengths of the anchor body have been tested. The research focuses on the analysis of waveforms registered by piezoelectric transducers attached in several locations of anchors.

2. Experimental investigation

The geometry of examined models of anchors is presented in Fig. 1. The specimens tested in experiment were steel rods embedded in concrete. Rods of a length of 1.5 m had a circular cross section with a diameter equal to 2 cm. Experiments were conducted for different bonding lengths: \(a_1 = 0\) cm, \(a_2 = 20\) cm, \(a_3 = 40\) cm and \(a_4 = 80\) cm. Material parameters of steel and concrete were equal to \(E = 210\) GPa, \(\rho = 7830\) kg/m³, \(v = 0.3\) and \(E = 26\) GPa, \(\rho = 2084\) kg/m³, \(v = 0.2\), respectively. The lagging thickness was equal to 4 cm for each anchor. In Figure 2 three experimentally tested specimens with different bonding lengths and the free rod are presented.

The geometry of examined models of anchors is presented in Fig. 1. The specimens tested in experiment were steel rods embedded in concrete. Rods of a length of 1.5 m had a circular cross section with a diameter equal to 2 cm. Experiments were conducted for different bonding lengths: \(a_1 = 0\) cm, \(a_2 = 20\) cm, \(a_3 = 40\) cm and \(a_4 = 80\) cm. Material parameters of steel and concrete were equal to \(E = 210\) GPa, \(\rho = 7830\) kg/m³, \(v = 0.3\) and \(E = 26\) GPa, \(\rho = 2084\) kg/m³, \(v = 0.2\), respectively. The lagging thickness was equal to 4 cm for each anchor. In Figure 2 three experimentally tested specimens with different bonding lengths and the free rod are presented.

Figure 1: Geometry of ground anchor

Figure 2: Photograph of experimentally tested specimens

Excitation and measurements of elastic waves were carried out by the device PAQ-16000D and piezoelectric transducers Noliac NAC2011. The actuator was located at the free end of the rod and receiving transducers were attached at different locations of the anchor. The function of excitation wave was obtained by multiplication of the sine function with a frequency of 80 kHz and the Hanning window.

The example of voltage signals for two bonding lengths are shown in Figure 3. Incident waves and reflections from the anchor body were identified and indicated. Information about the wave velocity and the time interval between the
incident wave and the reflected wave packet gives the opportunity to find a bonding length.

![Figure 3: Experimental voltage-time signals registered for anchors with two different bonding lengths: a) 40 cm; b) 80 cm](image-url)

3. Numerical analysis

Numerical investigation of elastic wave propagation in models of ground anchors were conducted in Abaqus/Explicit by the finite elements method. The FEM models were performed with the assumption of the rotational symmetry.

![Figure 4: Maps of velocity field propagated in specimen with bonding length equal to 80 cm at selected time instances: a) t = 1.25×10^4 s; b) t = 2.25×10^4 s; c) t = 3.5×10^4 s](image-url)

Anchors were discretized using 4-nodes finite elements of dimensions 1 mm × 1 mm. The size of the finite elements was adopted according to the length of the excited wave. Boundary conditions were assumed free on all edges. Excitation was executed by applying a time-dependent surface load on the free end of the anchor. In numerical models deterioration conditions of adhesion bonding between concrete and steel rod was not taken into account. Rayleigh damping model was applied and the mass-proportional coefficients for steel and concrete were adopted on the basis of calibration with experimental models.

The results of numerical investigations are presented in two formats: in the form of velocity maps of propagating wave in the volume of tested elements (Fig. 4) and in the form of acceleration-time signals (Fig. 5).

4. Conclusions

The paper presents experimental and numerical research on the possibility of the application of guided waves for the evaluation of the state of structural elements, impossible to be assessed during a standard visual inspection. Investigations were conducted for laboratory models of ground anchors with different bonding length. The accurate bonding length is a prerequisite providing high load capacity of the performed element because of the cooperation between steel and concrete over large surface. Results of the research proved the possibility of application of guided wave propagation in quality assessment of ground anchors.

References


Effectiveness of damage detection in 3-D structures using discrete wavelet transformation

Krzysztof Ziopaja
Faculty of Civil Engineering, Poznan University of Technology
Pl. Marii Skłodowskiej-Curie 5, 60-965 Poznań, Poland
e-mail: krzysztof.ziopaja@put.poznan.pl

Abstract

The method of damage identification using 1-D discrete wavelet transformation is proposed in the paper. 3-D structures subjected to static and dynamic action are considered. The damage identification is based on multiresolution analysis and structural response registered in a single measurement point. The experiments are numerically simulated. Measurement errors are introduced by means of a white noise simulation.

Keywords: discrete wavelet transform, damage detection, static and dynamic tests, bridges

1. Introduction

The problem of structural damage identification is one of the most important engineering issues. In a wide range of identification methods, the non-destructive techniques are the most interesting and promising. In identification methods, the structural response is analysed. The non-destructive methods based on optimization algorithms [6], on the dynamic response of the structure [2], artificial neural networks [9] or thermal conductivity [11]. In identification methods, the structural response is analysed. A powerful tool of signal analysis is wavelet transformation. It allows multiresolution analysis of 1-D or 2-D signals and due to its features is useful in detection of local disturbances of signals. Application of a wavelet transformation to problem of damage identification was presented in [4,5,7,8].

The focus of the work is the issue of the assessment of the technical conditions of bridges. Bridge constructions are of the three-dimensional complex beam and shell structures. Bridges are exposed to heavy variable loads which are of high amplitudes. The issue of damage detection in bridges is crucial due to high costs of abovementioned constructions. Some examples of damage identification are presented in [1, 3, 10].

2. Wavelet transformation as a tool in signal analysis

The mathematical basis of wavelet analysis was created in 1980s, Newland's, Chui's, Daubechies's or Mallat's publications and monographs introduced theoretical basis and provided a broad range of applications connected with wavelet analysis. In 1980s and during the next decades wavelet transformation became an extremely effective and advanced tool of the signal analysis of a wide range of problems.

The unquestionable and fundamental advantage of wavelet transformation is the fact that the analysing wavelet is characterized by a good localization in time and frequency (basic Sine and Cosine trigonometric functions applied by Fourier's analysis do not possess this quality). This key feature of wavelets is a great asset as for practical uses such as local singularities detection in signals, which is applied in the localization and detection of damage in engineering structures and defects in materials. In this article the discrete wavelet transform (DWT) is applied.

2.1. Discrete wavelet transform and multiresolution analysis

Wavelet coefficients \( d_{j,k} \) of the discrete wavelet transform \( W_{\psi} \) of a signal \( f(t) \) are defined as the scalar product of the signal and the wavelet function

\[
W_{\psi} f \left( \frac{1}{2^j}, \frac{k}{2^j} \right) = \langle f(t), \psi_{j,k} \rangle = d_{j,k} .
\]  

(1)

The basis function \( \psi \) (mother wavelet) creates the family of wavelets

\[
\psi_{j,k}(t) = 2^j \psi \left( 2^j t - k \right),
\]

where the scale parameter is defined as \( 1/2^j \), and translation parameter as \( k/2^j \) for \( j,k \in \mathbb{C} \).

The advantage of the discrete wavelet transform is the possibility of the representation of the signal \( f(t) \) as the sum of the smooth and detailed representation

\[
f(t) = \sum_{k \in \mathbb{Z}} a_{j,k} 2^j \phi(2^j t - k) + \sum_{k \in \mathbb{Z}} d_{j,k} 2^j \psi(2^j t - k) ,
\]

where \( \phi(t) \) is called scaling function (father) describe by an equation similar to (2). The equation (3) might be expressed in a different way: namely, as

\[
f(t) = A + \sum_{j=1}^{J-1} D_j .
\]

(4)

2.2. One dimensional decomposition

In the article, the efficient and simple Mallat’s algorithm of one dimensional decomposition of the signal \( f(t) \) is used

\[
f_j(t) = S_j + D_j + D_{j-1} + \ldots + D_m + \ldots + D_2 + D_1 ,
\]

(5)

where \( S_j \) means a smooth, constant part of the signal \( f(t) \), and the symbols \( D_{j-1} \) reflect the details consisting the local information about the signal. Parameter \( m=J-j \) indicates the level of the detail \( D_m \). In order to identify the damage, we will use only the decomposition details. The most detailed, so i.e. carrying a lot of detailed and local information is, usually, the \( D_1 \) detail. The function \( f_1(t) \), must be approximated by \( N=2^j \) discrete values. In the paper, orthogonal Daubechies wavelet family is applied.

3. Problem formulation

The main objective of the study is to present the ability to detect and locate damage in the numerical models of tree-dimensional structures. The model of the structure is as far as possible to simulate correctly a beam or beam-plate steel bridge structure or a structure of a composite steel-concrete type.

Figure 1: Model of a bridge structure

3.1. The procedure of damage identification

Assuming that there is damage in the construction (its definition was given in 3.2.), the methodology of the identification procedure is as follows (see also Figure 1):

- Concentrated load is moving on the deck of the bridge (quasi-static action)
- In the selected point of the main girder, or other structural element of the supporting structure, the geometric parameters (deflection, angle of rotation) were monitored
- The structural response is recorded in relationship with the moving load (so we get an influence function of vertical displacement or angle of rotation)
- The size of the signal depends on the load-shifting
- The signal is analyzed using one-dimensional DWT
- The signal transform correctly detects and localizes the damage.

The same way, damage identification procedure is used when the load action is dynamic. The main difference is the type of structural response (e.g. the amplitude of the vertical displacement or acceleration) and its correlation with the moving load.

3.2. Definition and model of the damage

Defects in structures (e.g. as a fatigue crack, corrosion) leads to stiffness reduction, increase of damping and decrease of natural frequencies. In the numerical examples a simple model of damage is used, most commonly used in the technical literature, which has the form of local stiffness reduction.

4. Numerical analysis

Selected types of bridge structures are considered. The numerical simulations were analyzed to detect internal damage, taking into account the influence of the local stiffness reduction, load position, influence of the number of measurements and the influence connected with the position of the measurement point. The influences of the white noise addition and detection of more than one defect are considered, too.

5. Concluding remarks

Numerical application of a discrete wavelet transformation to the damage detection proved that the method is very efficient. The main advantage of presented approach of damage detection is exertion of the response signal of "damaged" structure and measurement is realized in one selected point. The new approach presents the three-dimensional character of the structure work and an indirect loads action on the main analyzed structural elements.

References

Do we need the Aero number?

Piotr Józef Ziolkowski¹, Paweł Ziolkowski², Janusz Badur³

¹,² Energy Conversion Department, The Szewalski Institute of Fluid-Flow Machinery PASci
Fiszera 14, 80-231 Gdańsk, Poland
E-mail: pjziolkowski@wp.pl, jbi@imp.gda.pl
³ Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
E-mail: pjziolkowski@wp.pl

Abstract

The motivation behind this article is to explain a role of the Aero number (Ae - dimensionless spin friction coefficient) in universal modelling of electropumping effect of enhanced mass flow rate reported in micro- and nano-channels. The Ae number should be regarded as a ratio of torque friction coefficients to rotational viscosity coefficients. The recently discovered electropumping phenomena by De Luca et al. (J. of Chem. Phys., 138:154712-1–154712-10, (2013)) are based on nano-coupling of spin angular momentum with linear streaming momentum and the electro-osmotic draining effect. The theoretical background relies upon different spin slip of dipolar liquids when confined between hydrophobic and hydrophilic solid surfaces - this phenomenon combined with applied rotating electric field leads to pumping of liquid. In the paper a general formulation of spin slip condition is developed from primary principles. It leads to an explanation of the electro-pumping effect in nanochannels.

Keywords: dimensionless number, slip velocity, nanoflows, source of fluid spin

1. Introduction

Bonthuis [2] explored a mechanism for flow generation in fluid-filled nanochannels employing coupling between translational and rotational momentum. De Luca et al. [3-4] developed electropumping of water for rotating electric fields using nonequilibrium molecular dynamics (NEMD). De Luca et al. [3] proposed that slip boundary conditions for molecular spin and translational velocity play a key role and leads to the pumping effect. Owing to this inconvenient boundary condition confinement the spin angular momentum is observed and it conversion to a linear streaming flow.

In the work we develop a more general statement is developed for the molecular slip boundary condition. We start from a general form a boundary condition within a thin shell-like spin layer. We present results of a correct formulation of angular momentum boundary condition which allows for a general action similar to the rotating electric dipoles. We also include an example with a stationary flow of the Cosserat fluid through a pipe.

2. Angular momentum balance in the slip layer

A motion of slip layer particle is governed by its slip velocity \( v_s = v_T + v_A \) and the angular velocity \( \omega_s = \omega_T + \omega_A \). Inertia properties of the layer particle is described by the layer density \( \rho_s \) and the layer inertia tensor \( J_{ij} = \rho_s \epsilon_{ij} \), refer with: Fig. 1). According to the laws of classical mechanics, the second Euler law of motion is determined completely by two vector measures related with themselves by a Newton-like equation of motion: \( \frac{d}{dt} L = M \) - the vector \( L \) represents a whole resultant of the total angular momentum taken with respect to the common origin, and the vector \( M \) is an absolute vector defined the torque acting on the bodies. Both resultant vectors \( L \) and \( M \) have a contribution coming from moment of momentum and from the internal couples (Eqs. (1-2)) [1]:

\[
L = \int_A \int_B (1 + x \times s) dV + \int_S (l_s + x_s \times s_s) ds \quad (1)
\]

\[
M = \int_A \int_B \rho(c + x \times b) dV + \int_S (m_{(\alpha)} + x \times t_{(\alpha)}) ds + \int_S \int_{|\Delta A|} \rho(c_s + x_s \times b_s) ds + \int_{|\Delta S|} (m_{(\nu)} + x_s \times t_{(\nu)}) dl \quad (2)
\]

Figure 1: Outline of the Cosserat boundary layer and the pill-box balance domain

Well known parameters appear here in three-dimensional bulk continu, \( L, s \) are the bulk angular and linear momentum; \( c, b \) are the body couple and body force densities.

The main governing equations of electromagnetic phenomena in electrostatic regime, where the electric field \( E, \)
the displacement field $\mathbf{D}$, the polarization field $\mathbf{P}$, the electrical current density $\mathbf{J}_e$, and the electrical potential $\varphi$ all were averaged locally over their microscopic counterparts, are:

$$\text{rot} \mathbf{E} = 0$$
$$\text{div} \mathbf{D} = \text{div}(e \mathbf{E}) = \rho_{el}$$
$$\mathbf{D} = \varepsilon_0 \mathbf{E} + \mathbf{P} = e \mathbf{E}$$
$$\mathbf{J}_e = \sigma_{el} \mathbf{E}$$

(3)

Taking into account the Reynolds and Slattery transport theorem and Gauss-Ostrogradski and Stokes-Weathborn identities, after step-by-step calculations, we transform Eqn (1) to (1):

$$\int \int \int \mathbf{t} \cdot \mathbf{t} \, dV - \int \int \int \mathbf{t} \cdot \mathbf{n} \, ds$$

(4)

Let us introduce auxiliary unit versors:

$$\mathbf{e}_x = \{x \mid (y, z) \}, \quad \mathbf{e}_y = \{0, \mathbf{w} \}$$

(5)

that represent the slip velocity direction and the total angular slip velocity direction, respectively. We postulate, in some untypical models of turbulence, and the adherence coefficient $\nu_0$. Finally, a coefficient $c$ was postulated by [2].

4. Aero number

Let us consider a flow of fluid in a straight pipe of circular cross section of radius $R$, and $z$-axis of cylindrical coordinate system $(r, \varphi, z)$. In analogy to the Hagen-Poiseuille simplified flow, let the vectors of the translational velocity and the spin to be in the form $\mathbf{v} = v(r) \mathbf{e}_r$, $\mathbf{w} = \omega(r) \mathbf{e}_\varphi$. Let us define now the Aero number as dimensionless spin length (see Eqn (9)):

$$\text{Ae} = 1 - cR/\theta,$$

(10)

where $\theta$ is rotational viscosity coefficient.

Finally [1], if by $Q_{\theta \rightarrow r} = -(\pi / 8 \mu) p R^4$ we denote known Hagen-Poiseuille volume flux (the volume discharge) than taking $Q_{\text{om}} = \int v(r) dA$ one can obtain:

$$Q_{\text{om}} = Q_{\theta \rightarrow r} \left[ 1 - \frac{4}{k^2} \left( \text{Ae} + \frac{l_k(k)}{k^2 l_k(k)} \right)^{-1} \right]$$

(11)

It is obvious that the solution depends on value of the dimensionless number $\text{Ae}$ that physically should be interpreted as dimensionless spin friction coefficient. Flow enhancement, observed in nano-tubes, can be realised when $\text{Ae}$ is less than zero so $cR/\theta \geq 1$.

References


Dynamic behaviour of a Timoshenko periodic beam

Arkadiusz Żak\textsuperscript{1,8}, Marek Krawczuk\textsuperscript{2}, Wiktor Waszkowiak\textsuperscript{3}

\textsuperscript{1,2,3} Faculty of Electrical and Control Engineering, Gdańsk University of Technology

Narutowicza 11/12, 80-233 Gdańsk, Poland

e-mail: arkadiusz.zak@pg.gda.pl\textsuperscript{1}

Abstract

Periodic structures are characterised by unusual physical properties that are a direct consequence of their structural periodicity. Artificial periodic structures, mimicking those known from the nature, can be engineered in a special manner to possess desired properties. Engineering of periodic structures is a relatively new area of research, which still requires numerical and experimental investigations. The results presented in the work are focused on the dynamic behaviour of a Timoshenko beam of certain periodic properties. Various sources of structural periodicity were investigated by the authors such as: changes in elastic properties, geometry, as well as the presence of drill-holes. Their influence on selected dynamic characteristics, primarily natural frequencies as well as the presence of so-called frequency band gaps, were carefully studied by the use of the Time-Domain Spectral Finite Element Method (TD-SFEM).

Keywords: Timoshenko beam, periodic structure, TD-SFEM, dynamic behaviour, natural frequencies, frequency band gaps

1. Introduction

Periodic structures are characterised by unusual physical properties that are a direct consequence of their structural periodicity. Artificial periodic structures, engineered according to specific requirements, may have many potential and interesting applications including such as vibration isolators or acoustic filters \cite{1}. This is possible due to the presence of the so-called frequency band gaps in their frequency spectra, in which periodic structures strongly attenuate mechanical vibrations or propagating elastic waves \cite{2,3}. The location of the frequency band gaps within the frequency spectra of periodic structures can be controlled by parameters describing structural periodicity (number of features). On the other hand the width of the frequency band gaps is fully adjustable by the intensity of these features (changes in elastic properties, geometry, the radius of drill-holes, etc.).

Numerical investigations of the dynamic behaviour of periodic structures require special numerical tools that are capable to model precisely high frequency dynamic responses of such structures. One of such effective and robust numerical techniques is the Time-Domain Spectral Finite Element Method (TD-SFEM), as a numerical method widely used for studying wave propagation phenomena in engineering structures \cite{4}.

The results of numerical simulations presented in the work were obtained by the use of the TD-SFEM. They are related to the dynamics of a Timoshenko periodic beam. The influence of structural periodicity as well as the intensity of periodic features (elastic modulus, thickness, radius of drill-holes) on changes in the natural frequencies of the beam was thoroughly analysed.

2. Numerical model

Numerical analysis concerned a beam of certain periodic properties modelled according to the Timoshenko theory \cite{4}. Spectral beam finite elements defined based on the Chebyshev node distribution and approximation polynomials of the 5th order were used in this analysis. The displacement field within the beam spectral finite element was assumed according to \cite{5}.

It was also assumed that the beam under investigation (geometry: length \(L = 2000\) mm, thickness \(h = 10\) mm, width \(b = 10\) mm) was isotropic and made out of aluminium (material properties: \(E = 67.5\) GPa, \(\nu = 0.33, \rho = 2700\) kg/m\(^3\)). The beam was modelled by 200 spectral finite elements in total, while the number of degrees of freedom of the numerical model was 2,000. Periodic boundary conditions were applied.

All numerical calculations were carried out by the authors in the MATLAB\textregistered environment. The dynamic behaviour of the beam was studied as dependent on the structural periodicity (i.e. number of periodically spaced features along the beam length), while the intensity of the features was controlled by relative changes in: (1) elastic modulus \(\alpha\), where \(\alpha = 1 - E/E_0\), (2) beam thickness \(\beta\), where \(\beta = 1 - h/h_0\), as well as (3) relative radius \(a\) of drill-holes, where \(a = r/b\).

3. Numerical simulation results

Firstly the influence of the radius \(r\) of drill-holes was investigated on the natural frequency spectrum of the beam under consideration. It was assumed that in total 100 drill-holes were equally spaced along the beam length and drilled vertically. It can be clearly seen from Fig. 1 that both the presence of the drill-holes (periodic features) and their relative radius \(a\) (feature intensity) have a profound influence of the dynamic behaviour observed. The presence of wide frequency band gaps is well visible, while their widths are dependent on the location of the frequency band gaps is closely correlated with the structural periodicity \(N\) that is described in this case by the total number of drill-holes along the beam length and equal to 100.

Next the influence of relative elastic modulus change \(\alpha\) was studied, in a very similar manner to the case described previously. In this case it was assumed that at selected locations along the beam length, described by the same structural periodicity \(N = 100\), the elastic modulus was reduced over the length of the beam equal to 10 mm. Results presented in Fig. 2 show that contrary to the drill-holes the influence of the elastic modulus change \(\alpha\) is much more pronounced.

*The Authors of the work would like to gratefully acknowledge the support for their research provided by National Science Centre through the project UMO-2013/07/B/ST8/03741 Wave propagation in periodic structures. All results presented in this paper was obtained by use of software available at Academic Computer Centre in Gdańsk in the frame of a computational project.*
modulus, as a periodic feature of the beam, has a much smaller effect. Although the frequency band gaps are present around the same natural frequencies, as those seen in Fig. 1, their width is significantly smaller. This is despite the fact that relative elastic modulus change $\alpha$ spans over a much wider range of values up to 0.9.

Finally the influence of changes in the beam geometry was investigated. For that purpose the thickness of the beam was assumed as variable. Numerical analysis covered in this case relative thickness changes $\beta$. As before it was assumed that at selected locations along the beam length, described by the same structural periodicity $N = 100$, the thickness of the beam $h$ was reduced over the length of the beam equal to 10 mm. It is well seen from Fig. 3 that relative thickness change $\beta$ has a much stronger effect on the natural frequencies of the beam, similarly to the influence observed in the case of the drill-holes. However, it should also be noticed that in this case relative thickness change $\beta$ spans over a wide range of values up to 0.9.

4. Conclusions

Based on the results of numerical simulations presented the following general conclusions can be formulated by the authors:

1. Structural periodicity has a very profound influence on the natural frequencies of the beam under investigation and the presence of frequency band gaps.
2. The location of the frequency band gaps can be controlled by the number of the features responsible for structural periodicity $N$.
3. The width of the frequency band gaps can be controlled by the intensity of the features responsible for structural periodicity, i.e. $a$, $\alpha$ or $\beta$.
4. Due to strain/stress field discontinuity numerical models exhibit certain periodic features that can mask the dynamic behaviour of periodic structures under investigation.
5. The TD-SFEM is superior numerical technique to the classical FEM in terms of modelling dynamic behaviour due to higher orders of approximation polynomials and smaller numerical errors resulting.

References


Modelling of the viscoelastic properties of the technical fabric VALMEX

Krzysztof Żerdzicki¹, Paweł Kłosowski², Krzysztof Woźnica³

¹,² Faculty of Civil and Environmental Engineering, Gdańsk University of Technology
Narutowicza 11/12, 80-233 Gdańsk, Poland
email: krzzerdz@pg.gda.pl, klosow@pg.gda.pl
³ Institut National des Sciences Appliquées Centre Val de Loire, Laboratoire PRISME
88, boulevard Lahitolle F-18020 Bourges cedex, France
e-mail: krzysztof.woznica@insa-cvl.fr

Abstract

The analysis of the Burgers model used for the constitutive description of the technical fabric VALMEX viscoelastic behaviour is presented. It has revealed that the correctness of the identification depends strongly on the level of the immediate strain taken for calculations. Moreover, it was proved that the Burgers model can be used for modelling of the viscoelastic properties of the polyester reinforced PVC coated VALMEX fabric.

Keywords: architectural fabrics, viscoelasticity, Burgers model, VALMEX

1. Introduction

The research and engineering observation, conclude that the long-lasting behavior of technical fabrics fastened on a real construction can be defined as viscoelastic. For its identification, the rheological tests of creep or relaxation type are required. When modelling, the Argyris’ and more advanced Schapery’s non-linear approaches can be used (Ref. [1]). However, for the numerical modelling of the large scale civil engineering structures simpler models are more practical. Therefore, the aim of the presented study is to analyze the Burgers model used for description of the viscoelastic behavior of the technical fabric VALMEX.

2. Viscoelastic Burgers model

The Burgers model is a four parameter model representing the linear viscoelastic properties of a material using derivative relations. It is a series combination of the Maxwell and Kelvin-Voigt models [2]. The Burgers model for the uniaxial case of creep tests has the following formula:

\[ \varepsilon(t) = \sigma_0 + \frac{\sigma_0}{\eta_1} t + \frac{\sigma_0}{\eta_2} \left( 1 - e^{-\frac{t}{\eta_2}} \right), \]

where \( \sigma = \sigma_0 = \) constant is the stress level, \( E_1 \) is the instantaneous elastic modulus, \( E_2 \) is the delayed elastic modulus and \( \eta_1, \eta_2 \) are the viscous coefficients.

3. Experiments

The VALMEX material is a polyester reinforced PVC coated architectural fabric that has been used for 20 years as the canopy of the Forest Opera in Sopot.

In this research the creep tests with the stress level \( \sigma = 31.7 \text{ kN/m} \) were conducted separately for the warp and fill direction. The tests were carried out on the special strength machine dedicated to creep tests. This particular device has been designed at the Gdańsk University of Technology.

4. Parameters identification

Parameter identification was performed according to the procedure described in Ref. [3] and is presented in the graphical form in Fig.1.

Figure 1: Methodology of Burgers model parameters identification [2]

The \( t_0, t_1, t_2 \) denote particular time points of the creep test in succession, while \( \varepsilon_0, \varepsilon_1, \varepsilon_2 \) stand for the corresponding strain levels. Given the values evaluated from the experimental curve, the parameters \( E_1, E_2, \dot{\varepsilon}_0, \eta_1 \) were determined. Subsequently, these values have been used for the evaluation of the parameter \( \eta_2 \) from the nonlinear part of the curve using the least square method. The analysis of the obtained results revealed that the correctness of the identification depends strongly on the value of the immediate strain \( \varepsilon_0 \) taken for calculations. Therefore the sensitivity analysis with respect to immediate strain \( \varepsilon_0 \) and the limit time \( t_1 \) was performed.

The results of the identification and verification process for the warp direction (sample No. PELZ14) for three different levels of \( \varepsilon_0 \) are presented in Fig. 2 and Fig 3, respectively. The verification was accomplished by the calculation of the overall response of the material using the final values of the obtained parameters. It can be noticed that for all presented cases the
determination coefficient $R^2$ of both identification and verification detects a high value. However, the greater the immediate strain $\varepsilon_0$ is, the better correlation of identification in the range $t = 0 \div 2 \times 10^5 \text{ s}$ is observed. On the other hand, the higher the correlation coefficient is, the worse fitting of the simulation for the linear range is obtained. The same tendencies have been noticed for the second analyzed time range $t = 0 \div 4 \times 10^5 \text{ s}$. For the fill direction of the fabric an identical observation for both time ranges was made.

5. Conclusions

The obtained results can be used for practical purposes. Using the Burgers model (with the high value of $\varepsilon_0$), it is possible to describe material behavior in the first period of inelastic deformation, that will be well appreciated by engineers preparing the cut patterns of the fabric sheets.

The Burgers model with parameters detected for the lower value of $\varepsilon_0$ can reflect better the long-lasting behavior under long-term constant loading. It would be of the primary importance for the prediction of material performance built in a real construction and used through years (e.g. the VALMEX fabric).

Figure 2: Burgers model parameters for the limit time $t_i = 2 \times 10^5 \text{ s}$ and different $\varepsilon_0$ : identification of parameters

Figure 3: Burgers model parameters for $t_i = 2 \times 10^5 \text{ s}$ and different $\varepsilon_0$ : verification by numerical simulation

References

<table>
<thead>
<tr>
<th>Name</th>
<th>Page</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Markiewicz B.</td>
<td>401</td>
<td></td>
</tr>
<tr>
<td>Markiewicz I.</td>
<td>871</td>
<td></td>
</tr>
<tr>
<td>Markovic N.</td>
<td>385</td>
<td></td>
</tr>
<tr>
<td>Markowski T.</td>
<td>421</td>
<td></td>
</tr>
<tr>
<td>Maruszewski B.T.</td>
<td>849</td>
<td>O'Neill M.</td>
</tr>
<tr>
<td>Marynowski K.</td>
<td>499</td>
<td>Orlisz J.</td>
</tr>
<tr>
<td>Maslak M.</td>
<td>403</td>
<td>Osowski R.</td>
</tr>
<tr>
<td>Matiss I.</td>
<td>575</td>
<td>Ostopov O.</td>
</tr>
<tr>
<td>Matsui R.</td>
<td>151, 153</td>
<td>Ostopa-Luczkowska K.</td>
</tr>
<tr>
<td>Matveenko V.P.</td>
<td>869</td>
<td>Ostopowskoi P.</td>
</tr>
<tr>
<td>Mazurkiewicz L.</td>
<td>405</td>
<td>Oswalid R.</td>
</tr>
<tr>
<td>McPhedran R.</td>
<td>505</td>
<td>Oskinis G.</td>
</tr>
<tr>
<td>Melcer J.</td>
<td>407</td>
<td>Oziplbo M.</td>
</tr>
<tr>
<td>Menzel A.</td>
<td>781, 787, 803</td>
<td>Paco P.</td>
</tr>
<tr>
<td>Meronk B.</td>
<td>399, 409</td>
<td>Paczos P.</td>
</tr>
<tr>
<td>Mertuszka P.</td>
<td>405</td>
<td>Pajar M.</td>
</tr>
<tr>
<td>Mężyk A.</td>
<td>825</td>
<td>Pakowski R.</td>
</tr>
<tr>
<td>Misiekiewicz M.</td>
<td>399, 411, 413</td>
<td>Pakula M.</td>
</tr>
<tr>
<td>Miazio L.</td>
<td>43</td>
<td>Palac M.</td>
</tr>
<tr>
<td>Michałak B.</td>
<td>939</td>
<td>Pamin J.</td>
</tr>
<tr>
<td>Michalowska M.</td>
<td>263</td>
<td>Pandi P.</td>
</tr>
<tr>
<td>Michnej M.</td>
<td>533</td>
<td>Paruch M.</td>
</tr>
<tr>
<td>Michnik R.</td>
<td>265</td>
<td>Parus A.</td>
</tr>
<tr>
<td>Mierzwiczak M.</td>
<td>181, 711, 843</td>
<td>Patyk R.</td>
</tr>
<tr>
<td>Migórski S.</td>
<td>123</td>
<td>Pawelko P.</td>
</tr>
<tr>
<td>Mijušković O.</td>
<td>937</td>
<td>Pawlak Z.</td>
</tr>
<tr>
<td>Mikulski T.</td>
<td>725</td>
<td>Pawludy D.</td>
</tr>
<tr>
<td>Milewski S.</td>
<td>165, 171</td>
<td>Pawlowska S.</td>
</tr>
<tr>
<td>Minier J.</td>
<td>519</td>
<td>Pawlowska A.</td>
</tr>
<tr>
<td>Mishuris G.</td>
<td>227, 489, 611, 613, 681</td>
<td>Pawlowski P.</td>
</tr>
<tr>
<td>Misiurek K.</td>
<td>977</td>
<td>Pazdanowski M.</td>
</tr>
<tr>
<td>Misztalska E.</td>
<td>577</td>
<td>Pazdanowski M.J.</td>
</tr>
<tr>
<td>Mitura A.</td>
<td>789</td>
<td>Pazer E.</td>
</tr>
<tr>
<td>Mitushev V.</td>
<td>221, 233, 579</td>
<td>Peck D.</td>
</tr>
<tr>
<td>Mleczak A.</td>
<td>415</td>
<td>Pelczynski J.</td>
</tr>
<tr>
<td>Młyniec A.</td>
<td>557</td>
<td>Perek A.</td>
</tr>
<tr>
<td>Mochnacki B.</td>
<td>235</td>
<td>Perelmuter A.</td>
</tr>
<tr>
<td>Molenia M.</td>
<td>597</td>
<td>Perkowska M.</td>
</tr>
<tr>
<td>Mordich A.I.</td>
<td>69</td>
<td>Pernach M.</td>
</tr>
<tr>
<td>Morozov I.A.</td>
<td>851</td>
<td>Perzyńska K.</td>
</tr>
<tr>
<td>Morzyński M.</td>
<td>541, 629, 653</td>
<td>Pesavento F.</td>
</tr>
<tr>
<td>Moumni Z.</td>
<td>157</td>
<td>Pesetskaya E.</td>
</tr>
<tr>
<td>Movchan A.</td>
<td>503, 505</td>
<td>Petrov Y.</td>
</tr>
<tr>
<td>Movchan A.</td>
<td>503, 505</td>
<td>Petyr H.</td>
</tr>
<tr>
<td>Mozziński S.</td>
<td>671</td>
<td>Peczerski R.B.</td>
</tr>
<tr>
<td>Mrzygłód M.W.</td>
<td>649</td>
<td>Piancacka-Belkhayat A.</td>
</tr>
<tr>
<td>Mucha W.</td>
<td>417</td>
<td>Pichler B.</td>
</tr>
<tr>
<td>Muszka K.</td>
<td>561, 593</td>
<td>Peczysza E.</td>
</tr>
<tr>
<td>Muszyński A.</td>
<td>225</td>
<td>Pietko M.</td>
</tr>
<tr>
<td>Muzychkin J.A.</td>
<td>69</td>
<td>Piękarska W.</td>
</tr>
<tr>
<td>Myślecki K.</td>
<td>237</td>
<td>Pierini F.</td>
</tr>
<tr>
<td>Myśliński A.</td>
<td>125, 651</td>
<td>Pietraskiewicz W.</td>
</tr>
<tr>
<td>Nadolny A.</td>
<td>353</td>
<td>Pietruszczak S.</td>
</tr>
<tr>
<td>Nagórk W.</td>
<td>983</td>
<td>Pietrzak N.</td>
</tr>
<tr>
<td>Nakielni K.</td>
<td>11</td>
<td>Pietrzyk M.</td>
</tr>
<tr>
<td>Nalepka M.</td>
<td>419</td>
<td>Pilecki Z.</td>
</tr>
<tr>
<td>Naumenko K.</td>
<td>883</td>
<td>Piotrkowska-Wróblewska H.</td>
</tr>
<tr>
<td>Nawalaniec W.</td>
<td>221, 233</td>
<td>Pisarski D.</td>
</tr>
<tr>
<td>Nazarko P.</td>
<td>421</td>
<td>Piuskowski B.</td>
</tr>
<tr>
<td>Nepelski K.</td>
<td>423</td>
<td>Platek P.</td>
</tr>
<tr>
<td>Nikolić R.R.</td>
<td>329</td>
<td>Podgórski J.</td>
</tr>
<tr>
<td>Nitka M.</td>
<td>99</td>
<td>Pokorska L.</td>
</tr>
<tr>
<td>Noga S.</td>
<td>421, 425, 493</td>
<td>Pokusiński B.</td>
</tr>
<tr>
<td>Nowak M.</td>
<td>219, 599, 853, 857</td>
<td>Polak M.A.</td>
</tr>
<tr>
<td>Nowak J.</td>
<td>239</td>
<td>Polesek-Karczewsk S.</td>
</tr>
<tr>
<td>Nowak M.S.</td>
<td>629, 653</td>
<td>Polus L.</td>
</tr>
<tr>
<td>Nowak R.</td>
<td>707</td>
<td>Pomezan V.</td>
</tr>
<tr>
<td>Nowak Z.</td>
<td>853</td>
<td>Poiniski M.</td>
</tr>
<tr>
<td>Nowicki T.</td>
<td>183</td>
<td>Popik P.</td>
</tr>
<tr>
<td>Nowicki A.</td>
<td>217</td>
<td>Popov G.</td>
</tr>
<tr>
<td>Obara P.</td>
<td>347</td>
<td>Posiadała B.</td>
</tr>
<tr>
<td>Oesterle B.</td>
<td>27</td>
<td>Potoczek M.</td>
</tr>
<tr>
<td>Ogierman W.</td>
<td>581</td>
<td>Powalka B.</td>
</tr>
<tr>
<td>Okmiński A.</td>
<td>501</td>
<td>Pozorski J.</td>
</tr>
<tr>
<td>Okrajni A.</td>
<td>427</td>
<td>Pozorski J.</td>
</tr>
<tr>
<td>Okukl T.</td>
<td>429</td>
<td>Pozorski Z.</td>
</tr>
</tbody>
</table>
The book includes the entirety of contributions to PCM-CMM-2015 CONGRESS, SEPTEMBER 8 - 11, 2015, Gdańsk (Poland).

The PCM-CMM-2015 CONGRESS is a joint scientific event of:
• the 3rd Polish Congress of Mechanics (PCM)
• the 21st International Conference on Computer Methods in Mechanics (CMM)

The idea of a "Polish Congress of Mechanics" was firstly suggested in 2005 by the Polish Society of Theoretical and Applied Mechanics. The scope was intended to cover the whole range of problems of theoretical, experimental and computational mechanics as well as interdisciplinary issues, including industrial applications.

The 21st International Conference on Computer Methods in Mechanics continues the 44-year series of conferences dedicated to numerical methods and their applications to the mechanics-based problems. The meetings, organized biannually since 1973 provide a forum for presentation and discussion of new ideas referring to the theoretical background and practical applications of computational mechanics.

Both events - the 3rd Polish Congress of Mechanics (PCM) and the 21st Conference on Computer Methods in Mechanics (CMM) - are aimed at presenting current state-of-the-art research in the field of mechanics and providing a wide forum for discussion of new ideas on theoretical background, new technologies and computational methods in a vast domain of mechanics and related disciplines.