CMM-SolMech 2022 conference

S17 Computational and experimental advanced mechanics in engineering applications

List of abstracts

ID 4: *Modelling of plane framework structures made of laminated composite beams* – P. Szeptyński

ID 21: *Numerical analysis of cold-formed sigma-type steel beams reinforced with CFRP wraps and tapes* (poster) – K. Rzeszut, M. Dybizbański, I. Szewczak, P. Różyło

ID 34: *The influence of mounting place of FBG sensor in composite on the base of FEM simulations* – I. Jiregna, W. Kuś, W. Mucha

ID 41: *Assessment of the safety procedures during defusing of unexploded ordnance* – P. Nowak, T. Gajewski, P. Peksa, P. W. Sielicki

ID 43: *Numerical homogenization of multi-walled corrugated cardboard with imperfections* – D. Mrówczyński, T. Garbowski

ID 46: *Finite element analysis of a lower limb prosthesis made with the use of auxetic metamaterials* – A. Mrozek, T. Stręk

ID 56: *Identification of the self-stress states of tensegrity domes using a genetic algorithm* – M. Solovei, P. Obara

ID 69: *Metamodel of the laser heating process in rotary forming processes* – B. Mrzygłód, I. Olejarczyk-Wożeńska, M. Hojny, T. Dębiński, P. Marynowski

ID 76: *Saturated poroelastic half-space problem with axisymmetric cylindrical indenter* – K. Miura, M. Sakamoto

ID 84: *Experimental determination and numerical simulation of 3D printed 316L steel by FDM* – R. Roszak, M. Ziegenhorn, D. Schob, H. Sparr

ID 99: *Geometric modelling of rotary forming in virtual reality environment* – T. Dębiński, M. Hojny, P. Marynowski, B. Mrzygłód, I. Olejarczyk-Wożeńska

ID 101: *Numerical modelling of rotary forming with laser heating supported by multi-GPU acceleration* – M. Hojny, P. Marynowski, T. Dębiński, B. Mrzygłód, I. Olejarczyk-Wożeńska, T. Gądek

ID 107: *Influence analysis of environmental conditions on the surface layer of the rolling-sliding contact* (poster) – A. John, H. Bąkowski

ID 114: *Seismic analysis of scaffolding* – J. Bęc

ID 122: *Optimisation of a composite wood-concrete girder with fibre-composite reinforcement* – T. Socha, A. Denisiewicz, K. Kula

ID 131: *Experiment, modeling and simulation of sandwich plates with focus on parameter identification* – S. Hartmann, P. K. Dileep

ID 152: *Numerical modeling of compressed glulam joints with irregular shape of contact* – G. Sroka, A. Al Sabouni-Zawadzka, J. Pełczyński, W. Gilewski

ID 158: *Verification of blast mitigating potential of hedges in real applications* – T. Gajewski, R. Studziński, M. Malendowski, P. Peksa, W. Sumelka, P. W. Sielicki

ID 159: *Numerical simulations and laboratory tests structural system made of sigma type cold-formed profiles* – A. Denisiewicz, K. Kula, T. Socha

ID 160: *Numerical and experimental research on wooden joins with steel plate fasteners* (poster) – K. Kula, A. Denisiewicz, T. Socha

ID 178: *Protection of urban spaces* (keynote) – N. Gebbeken

ID 184: *Neural decision-making system of the mining sorter, activating the appropriate number of sorting nozzles* – M. Dudzik, J. Progorowicz

ID 193: *Probabilistic divergence application in structural reliability analysis* – M. Kamiński

ID 199: *Wave absorbing boundaries in dynamic analyses of wind turbines* – M. Steggewentz, O. Beilke, M. Stahl

ID 205: *Fracture energy of wooden members under contact explosion* – A. Szlachta, P. Sielicki, T. Grajewski

ID 214: *Discrete Wavelet Transform application for detecting damage in bars and nodes of truss structures* – A. Knitter-Piątkowska, O. Kawa

ID 219: *Experimental and numerical investigation of material and damage behaviour of 3D printed polyamide 12* – D. Schob, R. Roszak, K. Kotecki, D. Kurpisz, M. Ziegenhorn

ID 222: *Mechanical response of pre-buckled honeycomb structures manufactured additively* – M. Sarzyński, P. Płatek, P. Baranowski, I. Czernichowski

ID 244: *Identification of elastic material parameters based on digital image correlation results* – M. Nowak, P. Szeptyński, M. Maj

ID 245: *The dual-phase La2Zr2O7 + 8YSZ atmospheric plasma sprayed thermal barrier coatings: determination of thermal conductivity and thermal stress distribution* (poster) – A. Jasik

ID 261: *Combined experimental and analytical approach to determine limit load of a slender reinforced concrete column in operational conditions* – M. Drozdowska, A. Tomaszewska, M. Sondej

ID 271: *FEM simulation and optimization of fine-blanking process for aluminum alloy sheet* (poster) – Ł. Bohdal, L. Kukiełka, R. Patyk, M. Miksza, K. Kośka

ID 276: *Experimental-numerical methods for studying shock-induced fracture of rock* – P. Baranowski, M. Kucewicz, J. Małachowski, M. Pytlik

ID 299: *Dynamic contact analysis of a rotating annulus using reduced order system approach* – G. Akilesh, M. Pandey

MODELLING OF PLANE FRAMEWORK STRUCTURES MADE OF LAMINATED COMPOSITE BEAMS

Paweł Szeptyński[1]

[1] Cracow University of Technology, Poland

pawel.szeptynski@pk.edu.pl

A linear elastic model of a laminated composite beam is presented. It was proved that the discussed model - despite its simplicity - provides a description of deformation and internal forces distribution, which is in good agreement with numerical predictions as well as with experimental results. Analytical solutions derived for the discussed model enable formulation of single beam's stiffness matrix. Such matrices may be used for analysis of framework structures made of composite beams in the same way as in the case of regular Matrix Stiffness Method (MSM) of Structural Mechanics. Key problem in such an approach is proper definition of boundary conditions as well as interaction between any two structural elements, namely displacement compatibility and nodal equilibrium. The problems of formulation of boundary conditions and compatibility conditions as well as a computational example are discussed in the presented research.

NUMERICAL ANALYSIS OF COLD-FORMED SIGMA-TYPE STEEL BEAMS REINFORCED WITH CFRP WRAPS AND TAPES

Katarzyna Rzeszut[1], Maciej Dybizbański[1], Ilona Szewczak[2], Patryk Różyło[2]

[1] Institute of Building Engineering, Poznan University of Technology, Poland
[2] Lublin University of Technology, Poland

katarzyna.rzeszut@put.poznan.pl

The article presents selected issues related to the reinforcement of a cold-formed steel element with CFRP wraps and tapes. The first part of the article deals with the basics of cold-formed thin-walled steel elements and CFRP composite materials with special focus on steel market and its future prospects. Economic analysis was carried out based on literature, professional media and market trends review.
In the second part, the authors present original research on the strengthening of thin-walled sigma steel beams with the use of several numerical models prepared to represent the failure mechanism. The main aim of the study is verification and validation of FEM numerical model of steel beams made of thin-walled cold-rolled Blachy Pruszyński sigma profile retrofitted by CFRP wraps and tapes Sika CarboDur S. Very important part of this study is focused on investigation of influence of boundary conditions in FEM model developed in program Abaqus, included two ways of modeling so-called fork support using displacement constraints or non-deformable shell elements of the R3D4 type. The beams were modeled by means S4R shell or C3D8R solid finite element with linear or square shape function, and composite tape by S8R shell finite element. In the paper special attention was paid to the evaluation of the possibility to increase the load capacity with simultaneous limitation of displacements of the beams by appropriate localization of CFRP wrap or tape.

THE INFLUENCE OF MOUNTING PLACE OF FBG SENSOR IN COMPOSITE ON THE BASE OF FEM SIMULATIONS

Iyasu Jiregna[1], Wacław Kuś[1], Waldemar Mucha[1]

[1] Computational Mechanics and Engineering RMT4, Silesian University of Technology, Poland

ijiregna@polsl.pl

Composites reinforced with fibers are becoming increasingly popular as the market grows in demand for lightweight materials with high strength for specific purposes [1]. In composite materials, the real-time strain development due to external loads are the main factor to be detected correctly, and to do this, small-diameter fiber Bragg grating (FBG) sensors relatively to inclusions can be placed into the structure during manufacturing. On the other hand, the sensor placement needs special consideration as the measurements may depend on fiber-sensor location (distance to nearest fibers, angles between sensor and fibers). Therefore, this study deals with the strain measurements of glass fiber epoxy matrix and investigation of the relation of measured strain with a distance between FBG sensor and glass stiffener based on numerical methods. The typical glass-epoxy composite contains glass fibers with very small diameters, smaller or comparable to the sensor diameter, but in some applications (for example in civil engineering) glass-epoxy bars GFRP (glass fiber reinforced polymer) with diameters of few or even tens of millimeters [2]. The influence of sensor-fiber location on strain measurements is critical when active real-time decisions based on the strain levels must be taken. Furthermore, the structural health monitoring (SHM) process entails monitoring variation in material and geometric properties over time using periodically sampled response measurements from an array of sensors [3]. The SHM process involves a selection of the types, numbers, and locations of sensors [4]. Therefore, the investigation of the relationship between sensor placement and material properties is very important in SHM.

Acknowledgment. The work is partially financed from research subsidy of the Mechanical Engineering Faculty, Silesian University of Technology.

References

[1] Rajak DK, Pagar DD, Menezes PL, Linul E. Fiber-Reinforced Polymer Composites: Manufacturing, Properties, and Applications. Polymers. 2019 Oct;11(10):1667.
[2] Hamid NAA, Thamrin R, Ibrahim A, Hamid HA, Salleh N, Jamellodin Z, et al. Shear strength prediction for concrete beams reinforced with GFRP bars. In: MATEC Web of Conferences. EDP Sciences; 2017. p. 02013.
[3] Cremona C, Santos J. Structural Health Monitoring as a Big-Data Problem. Struct Eng Int. 2018 Jul 3;28(3):243-54.
[4] Testoni N, Aguzzi C, Arditi V, Zonzini F, De Marchi L, Marzani A, et al. A Sensor Network with Embedded Data Processing and Data-to-Cloud Capabilities for Vibration-Based Real-Time SHM. J Sens. 2018 Jul 2;2018:e2107679.

ASSESSMENT OF THE SAFETY PROCEDURES DURING DEFUSING OF UNEXPLODED ORDNANCE

Piotr Nowak[1], **Tomasz Gajewski**[1], **Piotr Peksa**[1], **Piotr W. Sielicki**[1]

[1] WILiT, Poznan University of Technology, Poland

piotr.nowak@doctorate.put.poznan.pl

75 years after the World War II (WWII) a great number of ordnance is found, recovered and destroyed by military specialists. During investments along the Polish coast explosive remnants of war are discovered in the proximity of urban areas. The cooperation between military and local authorities is essential for successful operation.
The aim of this study is to draw attention to the complexity of clearance operations of munitions sunk in sea or inland waterways and present novel hazard mitigation strategy for underwater ordnance disposal close to inhabited regions. Those naval weapons with high rates of explosive content demand an individual approach during neutralisation. A detailed risk assessment is an essential part of the planning process. The hazard factors for the shallow-water clearance are the underwater shock wave, fragmentation and seismic ground motion. As personal safety can be assured by evacuation and implementation of safety zones, the structural damage to buildings and bridges caused by seismic waves cannot be reduced in a simple way.
The presented measurements were recorded during the prepatation fase and the disposal of the 12000 lb Tallboy bomb. Detonation parameters of German WWII depth charges in the Piast Channel were the basis for a full scale risk evaluation before conducting an underwater neutralisation of the Tallboy. The obtained results were used by the Navy in the planning process and by civil authorities in the risk mitigation strategy. The registered data included pressure in time changes as well as seismic ground motion and seismic wave speed was scaled to a 3600 kg TNT explosion. The Tallboy disposal was registered by 12 seismic sensors. This data was used to calculate the efficiency of the deflagration proces. All data measured before and during disposal operations were used in order to maximise the safety of the civilians and navy divers. The disposal operation was conducted successfully at 13.10.2020 and according to our knowledge it was one of the biggest after WWII.

NUMERICAL HOMOGENIZATION OF MULTI-WALLED CORRUGATED CARDBOARD WITH IMPERFECTIONS

Damian Mrówczyński[1], Tomasz Garbowski[1]

[1] Department of Biosystems Engineering, Poznan University of Life Sciences, Poland

damian.mrowczynski@up.poznan.pl

In order to determine the parameters of the corrugated board, many laboratory tests should be carried out, i.e. bending, shear, torsional stiffness, resistance to edge crushing of samples and many others. The obtained parameters are often used to analytically determine the load capacity of corrugated cardboard packaging. For this reason, the correct calculation of these parameters is crucial in the design process of a corrugated board structure. In order to adequately reproduce the behavior of samples during laboratory tests, it is necessary to take into account the initial imperfections, which is especially important when compressing the layers of corrugated cardboard. Initial imperfections are most often caused by physical factors, such as the structure of cellulose fibers, local changes in humidity and temperature, as well as technological factors such as the process of producing paper rolls and the complicated process of gluing multi-layer corrugated cardboard. The work uses the finite element method to create the initial global stiffness matrix, which then undergoes static condensation and is directly used in the homogenization procedure. Using the homogenization method, it is possible to easily transform a complete model of a corrugated cardboard sample, i.e. a representative volumetric element (RVE), into a flat plate with effective parameters representing the membrane and bending stiffness. Such a transformation of the model allows to simplify and speed up the calculations without losing their accuracy. The work focuses on the numerical simulation of corrugated board tests, taking into account the imperfections of the cardboard layers, and also analyzed the impact of these imperfections on the values of the parameters obtained. The method was verified on the basis of experimental data and data contained in the literature. A high compliance of the computational model with the experimental data was obtained.

FINITE ELEMENT ANALYSIS OF A LOWER LIMB PROSTHESIS MADE WITH THE USE OF AUXETIC METAMATERIALS

Agata Mrozek[1], **Tomasz Stręk**[1]

[1] Faculty of Mechanical Engineering, Poznan University of Technology, Poland

agata.mrozek@doctorate.put.poznan.pl

Auxetic metamaterials allow to obtain unique properties at relatively low mass of the structure. Materials with negative Poisson's ratio could be characterized by some enhanced mechanical properties like higher energy absorption.
Dynamic properties of lower limb prosthesis significantly influences the quality of prosthesis usage. The insight into the overall dynamic response enable to improve existing structures. In this study the dynamic response of the foot prosthesis made with the use of auxetic metamaterials are analysed.
The proposed foot prosthesis models were created based on sketch of J-blade prosthesis, inspired by Cheetah Xtend model by Ossur using nTopology software. In models a re-entrant structure was included. Besides the geometrical parameters of the unit cells, the influence of unit cells orientation in relation to the given load were analysed.
To evaluate dynamic response of auxetic structure, mechanical impedance (Z) and vibration transmission loss (VTL) were analyzed using numerical simulations. Preliminary studies have shown the beneficial effect of using materials with a negative Poisson's ratio on the prosthesis response.
One of the structure parameters analysed was the orientation of the unit cells. Rotation of the re-entrant cell by 90 degrees caused a slight decrease of the first eigenfrequency value. Nevertheless, this allowed for a significant decrease in the acceleration amplitude. The study demonstrates that the application of auxetics structures could improve the dynamic properties of foot prosthesis as well as an appropriate adjustment of the geometric parameters of this structure.

IDENTIFICATION OF THE SELF-STRESS STATES OF TENSEGRITY DOMES USING A GENETIC ALGORITHM

Maryna Solovei[1], Paulina Obara[1]

[1] Faculty of Civil Engineering and Architecture, Kielce University of Technology, Poland

msolovei@tu.kielce.pl

The aim of this paper is to search for a self-equilibrium system of internal forces called the self-stress state. It is the most important feature of tensegrity structures. The identification of self-stress states leads to finding a tensegrity form (form-finding methods). Over the last years, several studies have been made on the form-finding methods of tensegrity systems. Some of them using a genetic algorithm.

The study includes the Geiger dome, Levy dome, Kiewitt dome, and their modifications. The geometries of these structures are known therefore the characteristic features can be identified by spectral analysis of the truss matrices (compatibility matrix and stiffness matrix with the effect of self-equilibrated forces). The main problem in the case of tensegrity domes is existing of multiple states of self-stress. None of them ensures stability and none correctly identifies the type of elements (that is, what is a strut and what is a cable). Only the superposition of these states leads to obtain an appropriate self-equilibrium system that ensures the stability of the structure.

In this paper, the identification of an optimal equilibrium system for tensegrity domes characterized by multiple states of self-stress is carried out. The first stage includes verification of tensegrity systems. A singular value decomposition (SVD) of the compatibility matrix is performed. Next, a genetic algorithm is adopted to uniquely define a single integral feasible set of self-equilibrium forces. The design variable is defined using only the constraint of the element types. The accuracy of obtained results is checked by overall equilibrium using a spectral analysis of stiffness matrix with the effect of self-equilibrated forces. Additionally, an equilibrium in individual nodes is checked. In all analysed domes, the single optimal self-stress state is obtained.

The numerical analysis is provided by using the calculation procedure written in the Mathematica environment.

METAMODEL OF THE LASER HEATING PROCESS IN ROTARY FORMING PROCESSES

Barbara Mrzygłód[1], Izabela Olejarczyk-Wożeńska[1], Marcin Hojny[1], Tomasz Dębiński[1], Przemysław Marynowski[1]

[1] AGH University of Science and Technology, Poland

mrzyglod@agh.edu.pl

Simulations of rotary forming processes with material heating by laser beam have not been widely used so far, due to limitations related to simulation times. The appearance on the market of software using GPUs to accelerate calculations, the widespread access to new design techniques using virtual reality (VR) systems, as well as the popularity of artificial intelligence (AI) decision-control systems in line with the Industry 4.0 concept, creates new opportunities to design new technologies for the aerospace industry.
The aim of the work is to develop a metamodel of laser heating in rotary forming processes for a virtual reality system supporting the design of new technologies for the aerospace industry.
The metamodel will be developed using advanced artificial intelligence methods based on experimental data from the laser heating process preceding the rotary forming process and simulation results from the numerical model of the rotary forming process with material reheating by laser beam developed by the co-authors.

SATURATED POROELASTIC HALF-SPACE PROBLEM WITH AXISYMMETRIC CYLINDRICAL INDENTER

Kotaro Miura[1], **Makoto Sakamoto**[2]

[1] Department of Systems Design Engineering, Seikei University, Japan
[2] School of Medicine, Niigata University, Japan

k-miura@hirosaki-u.ac.jp

We considered the saturated poroelastic half-space problem assumed to be applied distributed load by the axisymmetric cylindrical indenter. In this study, we utilized the axisymmetric displacement functions and applied the Laplace and the Hankel transform to the governing equations. Finally, the dual integral equations in the Laplace transform domain were reduced to an infinite system of simultaneous equations by expressing a normal contact stress at the surface of half-space as an appropriate series. It was necessary to utilize numerical inversion of the Laplace transform to obtain the numerical results in real-time domain. The numerical inversion method to be used was Talbot's method. Numerical results of the degree of consolidation displacement of the cylindrical indenter were obtained to demonstrate the comparison between the results for that of the previous other studies. The present results were in good agreement with the numerical results of specific study. We will consider the saturated poroelastic multi-layered problem and apply the semi-analytical solution to the indentation tests of biomaterials to obtain the mechanical properties of those materials.

EXPERIMENTAL DETERMINATION AND NUMERICAL SIMULATION OF 3D PRINTED 316L STEEL BY FDM

Robert Roszak[1], Matthias Ziegenhorn[1], Daniela Schob[1], Holger Sparr[1]

[1] Mechanical Engineering, BTU Cottbus Senftenberg, Germany

roszak@b-tu.de

In order to characterise the material and damage behaviour of additively manufactured 316L steel under quasi-static load and to implement it in a numerical model, experiments under quasi-static load as well as microstructural investigations were carried out. FDM was used as the manufacturing process. For the classification of the material behaviour, quasi-static cyclic tests with holding times as well as tensile tests were performed. X-ray refraction and computed tomography (CT) were used to investigate the damage behaviour. The Chaboche model, which has already been applied for metallic materials under thermomechanical loading, served as the basis for the selection of the numerical material model. The same procedure was used for the selection of the damage model, where the Gurson-Tvergaard-Needleman (GTN) model was chosen, which was already used for porous metallic materials. The Chaboche model shows very good agreement with experimental results. Furthermore, the coupling with the GTN model allows a very good modelling of the damage behaviour. Finally, it could be shown that the selected models are suitable to simulate the material and damage behaviour of 3D printed 316L steel.

GEOMETRIC MODELLING OF ROTARY FORMING IN VIRTUAL REALITY ENVIRONMENT

Tomasz Dębiński[1], Marcin Hojny[1], Przemysław Marynowski[1], Barbara Mrzygłód[1], Izabela Olejarczyk-Wożeńska[1]

[1] Department of Applied Computer Science and Modeling, AGH University of Science and Technology, Poland

debinski@agh.edu.pl

As part of the lecture, a virtual reality (VR) system dedicated to supporting the design of new rotary forming technologies with heating the material with a laser beam will be presented. The basis of the developed solution are numerical algorithms allowing for geometric modeling of the material flow depending on the designed configuration of the rollers and their trajectory. In addition, the developed methods will be discussed, allowing: the coupling of the virtual copy of the machine with its real (industrial) counterpart and the computing engine that uses the GPU processors of NVidia graphics cards to accelerate the calculations. The tests carried out have shown that the developed new approach using high-performance numerical calculations and the capabilities of virtual reality systems allows for effective design and control of the technological process in industrial conditions.

ID 101

NUMERICAL MODELLING OF ROTARY FORMING WITH LASER HEATING SUPPORTED BY MULTI-GPU ACCELERATION

Marcin Hojny[1], Przemysław Marynowski[1], Tomasz Dębiński[1], Barbara Mrzygłód[1], Izabela Olejarczyk-Wożeńska[1], Tomasz Gądek[2]

[1] Department of Applied Computer Science and Modeling, AGH University of Science and Technology, Poland
[2] Łukasiewicz - Poznań Institute of Technology, Poland

mhojny@agh.edu.pl

Numerical simulations of rotary forming processes with heating the material with a laser beam have not been widely used so far due to the limitations associated with simulation time. The appearance on the market of solutions using the CUDA architecture of multi-core processors (mainly graphics cards) for acceleration of calculations, widespread access to new interactive design techniques using virtual reality (VR) systems, as well as the popularity of artificial intelligence (AI) decision-control systems compatible with the concept of Industry 4.0, creates new opportunities to design new technologies. The main goal of the design and implementation work was to develop an advanced numerical model of the rotary forming process with heating the material with a laser beam (stationary or moving beam). The completed experiments and simulations of the 316L stainless steel semi-product spinning process showed that the developed numerical model allows the manufacturing process to be effectively engineered and controlled in industrial conditions.

INFLUENCE ANALYSIS OF ENVIRONMENTAL CONDITIONS ON THE SURFACE LAYER OF THE ROLLING-SLIDING CONTACT

Antoni John[1], **Henryk Bąkowski**[2]

[1] Mechanical Engineering, Silesian University of Technology, Poland
[2] Transport, Silesian University of Technology, Poland

antoni.john@polsl.pl

The paper presents a 3D and 2D model of the surface topography developed on the basis of tests of an association reproducing real operating conditions.
The influence of various operating conditions, i.e. the presence of water or solid lubricant in the wheel-rail contact, was determined. In order to determine the influence of a single factor on such a complex system as the wheel-rail system, tests on a laboratory stand were used. The special feature of laboratory testing is that it is more possible to control the direct influence of one selected factor than it is under real conditions. Both rigid and deformable materials were used in the model tests.
The presented results in the form of stress and strain distribution and the depth of their maximum values will be compared with the results of metallographic examinations of the surface layer. This makes it possible to determine the wear mechanisms and predict the wear character.
In this way, the influence of the environment on the impact (its destructive character) of three factors (presence of water, solid lubricant and no lubrication - dry contact) on the condition of the surface layer is determined and, based on the depth of occurrence of maximum stresses of the wear mechanism (abrasive, adhesive or surface fatigue) is predicted.
The depth at which the maximum values of stress and strain are located is closely related to the thickness of wear debris. Therefore, it is possible to formulate a thesis that the type and intensity of wear and prognosis of the durability of a rolling-sliding combination can be determined on the basis of stereological features of wear debris.

SEISMIC ANALYSIS OF SCAFFOLDING

Jarosław Bęc[1]

[1] Department of Structural Mechanics, Lublin University of Technology, Poland

j.bec@pollub.pl

Scaffolding is a lightweight temporary structure used mainly at construction sites. Since natural frequencies of such structures are low and first ones are usually in the range of 2-4 Hz, they are prone to dynamic loads. These dynamic actions are generated by an equipment present at the site, by moving workers, wind action, nearby traffic or other vibration propagated via soil from the machines in the vicinity of the scaffolding. Special types of dynamic actions are seismic and paraseismic activities which will be presented in this paper. These actions are generated by a massive motion of earth surface as the result of earthquakes, or in case of paraseismic action, as the result of ground motion induced by the mine collapse. In both cases these vibrations are transmitted to the scaffolding structure as a whole through the ground supports, as well as through the anchors fixed at the neighboring main building structure.

In the years 2016-2018, 120 facade frame scaffoldings in Poland were analyzed during the ORKWIZ project realized by the consortium of research groups conducted by LUT. The main subject of this project was scaffolding and workers safety. Various parameters of structures were measured and analyzed including dynamic properties of scaffolding structures. Numerical models of all analyzed scaffoldings were prepared in Autodesk Simulation Mechanical 2013. The numerical models and dynamic parameters of these structures such as natural frequencies and values of logarithmic decrement of damping were verified on the basis of measurements in-situ. It had been observed that these parameters depend not only of the scaffolding size and dimensions, but one of the most influential factors is the accuracy and quality of scaffolding structure anchorage at the main building structure.

Various scaffolding models of different dimensions and dynamic characteristics have now been subjected to the numerical analysis. Seismic action has been analyzed according to the procedures of Eurocode 8 by using time-history representation. Paraseismic activities have also been analyzed. Both of these actions generate mostly horizontal vibrations, however the key difference is the duration of the dynamic perturbation. Earthquakes generate prolonged disturbances lasting for several seconds while paraseismic vibrations of human activity origin are of almost impulse character. The results obtained with direct integration analysis of selected scaffolding structures subjected to seismic or paraseismic actions have been compared. The provided analysis has showed which scaffolding structures are the most endangered by such types of actions.

OPTIMISATION OF A COMPOSITE WOOD-CONCRETE GIRDER WITH FIBRE-COMPOSITE REINFORCEMENT

Tomasz Socha[1], Arkadiusz Denisiewicz[1], Krzysztof Kula[1]

[1] Institute of Civil Engineering, University of Zielona Góra, Poland

t.socha@ib.uz.zgora.pl

This paper presents an analytical and numerical approach to the problem of optimisation of a composite girder consisting of three components: a wooden truss, a concrete slab on a compression upper chord and a composite reinforcement for the tension lower chord. Wood, despite its many advantages (relatively low weight, relatively high compressive and tensile strength and ease of processing and assembly), also has disadvantages. It has many structural defects, and it is a heterogeneous and anisotropic material. Its properties depend on many factors such as humidity, temperature and long-term loading. The strength of wood decreases due to structural defects and/or inappropriate fibre orientation. Despite its high chemical resistance, it is attacked by various pests or fungi. These disadvantages can be eliminated by reinforcing wooden structures with fibre-composites, especially in the tension zone of beams. The technological development of FRP composites leads to their wider use in the strengthening of wooden elements. It provides greater load-bearing capacity and stiffness of the elements. A similar coefficient of thermal expansion to wood allows to both materials to be fully bonded. The reinforcement of the tension zone with fibre-composite can be additionally exploited by placing a concrete slab in the compression zone. This composite structure has many advantages. Unfortunately, a major problem is the correct and optimal combination of the parameters of the timber girder, the fibre-composite reinforcement and the concrete slab. A wooden truss with a bar system optimised for stiffness at the lowest possible girder height was adopted as the main structural element. The truss bars were connected using metal plates. The bottom chord was reinforced with CFRP tape, while the top chord had a concrete slab on top of OSB plate. The cross-sections and material parameters of these elements were optimised taking into account several factors (bearing capacity, weight, costs). High-performance concrete was used to reduce the weight and thickness of the concrete slab. The calculations were made assuming that all elements were fully joined without displacements. The ABAQUS FEM software was used for the calculations. These types of composite girders can be used in bridge and ceiling structures, even for long span.

EXPERIMENT, MODELING AND SIMULATION OF SANDWICH PLATES WITH FOCUS ON PARAMETER IDENTIFICATION

Stefan Hartmann[1], **Pranav Kumar Dileep**[1]

[1] Institute of Applied Mechanics, Clausthal University of Technology, Germany

stefan.hartmann@tu-clausthal.de

Sandwich plates, which consist of outer metal face sheets and an inner fiber-reinforced plastic layer, are of special interest in forming technology. In this paper, the process of conducting experiments to characterize the metal properties as well as the fiber-reinforced plastic components - made of a woven fabric - is studied in terms of identifying the material parameters at hand. A special focus is on the unique determination of the parameters, leading to a model reduction of a model of viscoplasticity at large deformations. Here, the concept of identifiability is chosen to investigate whether the parameters are uniquely identifiable. In addition, the parameters of a model of orthotropic hyperelasticity are determined from micro-CT data by numerical homogenization. Uncertainties arise in all parameter identification steps. These are used with the aid of Gaussian error propagation for uncertainty quantification of the resulting forming process simulation. In other words, the entire process of experiment, modeling, and identification is used to estimate the uncertainties of the simulation resulting from the material parameter determination.

NUMERICAL MODELING OF COMPRESSED GLULAM JOINTS WITH IRREGULAR SHAPE OF CONTACT

Grzoegorz Sroka[1], Anna Al Sabouni-Zawadzka[1], Jan Pełczyński[1], Wojciech Gilewski[1]

[1] Faculty of Civil Engineering, Warsaw University of Technology, Poland

j.pelczynski@il.pw.edu.pl

Timber connections, including the joints in glued laminated timber, can be divided into two categories: indirect and direct [1]. In indirect connections the loads are carried through additional elements, such as metal plates, nails, glue [2]. The load carrying capacity of direct joints results mainly from the shape of the connected members, they can be additionally stabilized with an additional element (e.g. dowel, nail). Automated production makes it possible to fabricate direct woodworking joints with complicated geometry, such as the ones produced with a CNC milling machine [3]. In this paper, a computational analysis of one of the joints described in [3] is presented. The aim of this study is to analyze selected contact models using the finite element method [4,5] and compare them with experimental results [3].
Due to complex geometry of the joined members and their contact area, the analyzed connection was divided into finite elements with gradually changing mesh density. Numerical simulations were performed in Abaqus [5], using a 3D model within the linear elasticity theory. An orthotropic material model was applied [4]. 1st and 2nd order cuboid and tetrahedral finite elements were used. Convergence of the solution was determined based on several meshes.
In the analysis, various contact models were tested. Bilateral tie constraints were applied using the option Constraint/Tie. Unilateral constraints were introduced in various variants, based on two types of the Contact Pair option: General Contact and Surface-to-surface Contact. The lower contact area was treated as Master, the upper - as Slave. The contact properties are determined by sliding formulation and friction coefficient.
The results were analyzed within displacements, stress and contact pressure. Selected displacements were compared with experimental results described in [3].

References

[1] Thelandersson S, Larsen HJ, editors. Timber engineering. New York: J. Wiley; 2003
[2] Bedon C, Fragiacomo M. Numerical analysis of timber-to-timber joints and composite beams with inclined self-tapping screws. Composite Structures 2019;207:13-28
[3] Kromoser B, Braun M. Towards efficiency in constructive timber engineering - design and optimization of timber trusses. Proc. of the IABSE Congress 2019, New York, USA, 2019
[4] Al Sabouni-Zawadzka A, Gilewski W, Pełczyński J. Perpendicular to grain stress concentrations in glulam beams of irregular shape - finite element modelling in the context of standard design. International Wood Products Journal 2018;9:157-63
[5] Abaqus Version 6.13. Simulia. ABAQUS User Subroutine Reference Guide. Providence, RI, USA: Abaqus; 2013.

VERIFICATION OF BLAST MITIGATING POTENTIAL OF HEDGES IN REAL APPLICATIONS

Tomasz Gajewski[1], Robert Studziński[2], Michał Malendowski[1], Piotr Peksa[1], Wojciech Sumelka[1], Piotr W. Sielicki[1]

[1] Institute of Structural Analysis, Poznan University of Technology, Poland
[2] Institute of Building Engineering, Poznan University of Technology, Poland

tomasz.gajewski@put.poznan.pl

Terrorists are attracted by open-air gatherings of people, especially concerts or out-door sport events, such as olympics etc. As recent scientific literature has shown, the tree hedges have the potential to give the protection of the blast waves from moderate charges from improvised explosive devices in such open-spaces events. In this study the comprehensive study on selected hedge trees was conducted.

In this study, the experiments on two thuja hedges (Thuja occidentalis 'Smaragd') were performed in full-scale field tests. Several distances behind the hedges were considered and several positions along the hedge, in which the overpressures were measured at the level of the chest of an adult male. The hedge wall was about 2.0m long and 0.55m thick. The nominal height of each tree was 1.5 m. The trees were planted in two staggered rows. The explosive used was 5kg of trinitrotoluene with a rectangular shape. Four detonations were registered with six pressure gauges. Also, two cameras recorded the in-plane and out-of-plane motion of the hedge. The plots of motion of selected points of the hedge wall were determined in time (out-of-plane movement).

The moderate reduction of overpressure peak and overpressure impulse were obtained and were compared with its counterparts, i.e. with cases in which there was no hedge wall. Also, the research results were compared to other studies. In our study, the overpressure peak reduction obtained was even 22 %. It was confirmed that the thuja hedge are adequate for protection in multiple bombing attacks since after several shots there was a small amount of removed needless from each tree. The measurements shown that thuja hedge wall have the potential to substantially reduce a harmfulness of moderate size improvised explosive devices, such as a suitcase, a backpack or a handbag bomb due to blast wave load.

NUMERICAL SIMULATIONS AND LABORATORY TESTS STRUCTURAL SYSTEM MADE OF SIGMA TYPE COLD-FORMED PROFILES

Arkadiusz Denisiewicz[1], Krzysztof Kula[1], Tomasz Socha[1]

[1] Division of Structural Mechanics, University of Zielona Góra, Poland

a.denisiewicz@ib.uz.zgora.pl

The paper presents advanced numerical simulations and laboratory tests of structural systems of steel beams made of sigma-type cold-formed profiles braced by platforms. The influence of deck type on the load carrying capacity of the structural system was investigated at different beam lengths from 3m to 6m. Steel lattice decks and decks made of thick chipboard panels type P4 and P6 were analysed. The influence of perforation on the load capacity of the beams was also studied. The technological perforation were placed on the web and on the side flanges of the top and bottom chords of the beam. This is a very significant issue from the engineering practice point of view, since Eurocode 3 does not take into account the influence of such imperfections on the load capacity of structures made of cold-formed profiles. The tested structural system consisted of three beams attached by steel connectors and bolts to a rigid body and decks positioned on the upper flanges of the beams. The tested structural system consisted of three beams attached by steel connectors and bolts to a rigid body and decks positioned on the upper flanges of the beams. The laboratory testes were performed for providing data for the validation of the numerical models. Two structural systems ware tested, one with a single steel lattice deck placed centrally on the beams and the second with a chipboard deck fixed over the entire span. In the case of the steel lattice deck, it was not possible to fix it for the entire span during the laboratory tests because of the way the load was transferred from the testing machine to the system under test. The laboratory tests were carried out on natural scale specimens 3m and 6m on an INSTRON 8804 testing machine with an additional actuator. Numerical simulations have been performed in the SIMULIA ABAQUS system. Numerical investigations carried out on a validated model made it possible to analyse the structural system at any span and with the use of upper flange bracing by the deck plates over the whole span, which corresponds to the real conditions. Numerical simulations were carried out taking into account physical and geometrical nonlinearities of the model, as well as the contact between deck plates and upper flanges of beams. The investigations give the possibility to determine the areas of bearing capacity as a function of the span of the structural systems.

NUMERICAL AND EXPERIMENTAL RESEARCH ON WOODEN JOINS WITH STEEL PLATE FASTENERS

Krzysztof Kula[1], Arkadiusz Denisiewicz[1], Tomasz Socha[1]

[1] Uniwersytet Zielonogórski, Poland

k.kula@ib.uz.zgora.pl

This paper presents advanced numerical simulations and laboratory tests of wooden joins using metal plate fasteners. The metal plate fastener is a modern element used as a connector in timber structures. It is a steel plate (galvanised or stainless steel) with spikes embossed on one side. The wooden parts are permanently connected by pressing the spiked plate together using high pressure presses (min. 18 tons). The main advantage of such fasteners is their ability to join lumber pieces together, guaranteeing high strength joints with consistent and predictable strength. Provided that the joint is made correctly, according to the technology. In engineering practice, one may encounter various execution defects in such joints, which are the result of geometrical inaccuracies or the use of inappropriate technology.

The study examined specimens with different joint defects: a non-axially fastened plate, a plate fastened manually with a hammer instead of a hydraulic press, a plate additionally fastened with nails, and a reference joint made correctly. Laboratory tests were carried out on life-size specimens. The behaviour of the joint in uniaxial tension was investigated. The tests were carried out using an INSTRON 8804 testing machine. The main objective of the laboratory tests was to provide data for the validation of numerical models, which were performed in the SIMULIA ABAQUS system. The numerical simulations allowed analysing the influence of individual joint defects on its strength. Simultaneously, it was possible to study the influence of the intensity of a given joint defect on the load capacity of the whole joint. For example, the influence of the size of the angle of deviation of the spiked plate axis from the joint axis. The numerical simulations were carried out taking into account the nonlinearities of the model in the physical and geometric ranges and taking into account the contact between the steel plate fasteners and the wooden elements.

PROTECTION OF URBAN SPACES

Norbert Gebbeken[1]

[1] Research Center RISK, University of the Bundeswehr Munich, Germany

norbert.gebbeken@unibw.de

Protection of urban spaces against vehicle attacks has become a big issue in Europe. The European Commission has recently awarded appr. 100 Mio Euros for a programme on the security of urban spaces (research and innovative best practise examples). The bollard industry has launched different initiatives (advertisement, code development, etc.) telling decision makers; you have a problem, we have the solution. But society and architecture sociologists oppose that installing bollards at outstanding urban public spaces restricts freedom and is against a city planning philosophy that is based on urbanity. Consequently, we have an interesting debate where we protective structures engineers are challenged. The paper and the presentation will deal with the following aspects: Risk study, societal debate, barrier solutions - from standard bollards to innovative barriers. Finally, a case study and a best practise example will be demonstrated.

NEURAL DECISION-MAKING SYSTEM OF THE MINING SORTER, ACTIVATING THE APPROPRIATE NUMBER OF SORTING NOZZLES

Marek Dudzik[1], Jakub Progorowicz[2]

[1] Faculty of Electrical and Computer Engineering , Cracow University of Technology, Poland
[2] Research and development department, Comex Polska Sp. z o.o. , Poland

marek.d@comex-group.com

The sorting process using vision systems is complex and multidisciplinary. Therefore, the difficulties occurring during it are wider than the issues of Binary Classification.
One of these difficulties is the appropriate selection of nozzles activating the stream of sorting air. The essence of the proper switching of the nozzles is clearly responsible for the quality of sorting. Incorrect selection of the activation of nozzles for a sorted object causes that there are situations in which the stream of sorting air affects both the sorted object and other objects located in its vicinity. In such situations, there is an undesirable effect of this stream on objects that are also sorted in subsequent iterations of the sorting process. Such situations are important because, if they are not properly identified and neutralized, they can lead to ineffective operation of the sorter. As a result, the sorter cannot be integrated in the production process of a specific mining product.
The article presents a neural network which is helpful in the process of neutralizing the influence of the sorting stream on the elements in the vicinity of the sorted object. Data for the network was collected directly from the sorter working with the actual mining material. The proposed network model is the result of the optimization of its structure. The obtained convergence of the results with the actual data is over 99%, which in real conditions is the desired effect of the sorter system operation due to the rounding to the intiger data type.

PROBABILISTIC DIVERGENCE APPLICATION IN STRUCTURAL RELIABILITY ANALYSIS

Marcin Kamiński[1]

[1] Lodz University of Technology, Poland

marcin.kaminski@p.lodz.pl

Probabilistic divergence (relative entropy) has been invented to measure a distance in-between different probability distributions, so that it should be useful in reliability assessment of engineering structures. The main aim of this work is numerical analysis based on the Stochastic Finite Element Method to study an interrelation of the few available relative entropies models with the FORM reliability index. Such an analysis is necessary to check whether different relative entropies models can return numerical values close to these existing in the designing engineering codes. Alternatively, some calibration procedures would be recommended. A distance of two probabilistic distributions is analyzed here in addition to structural resistance and to its effort, where both functions are assumed to have Gaussian distributions and to be uncorrelated at all. It should be mentioned that this PDF can be further relatively easily replaced with some other technically important probability distribution functions.

The Stochastic Finite Element Method (SFEM) employed in this study would be based on three alternative probabilistic computational techniques - Monte-Carlo simulation, generalized iterative stochastic perturbation technique as well as the semi-analytical approach. They would be used to calculate the first two probabilistic moments of structural response and they represent in turn statistical approach, Taylor expansion of the given order as well as analytical integration of the response functions. As it is well known, the existing engineering software does not allow for a direct FEM-based reliability index of neither existing nor newly designed structures. Deterministic structural problems are discretized and solved here with the use of commercial FEM environments of the package Autodesk ROBOT and of the system ABAQUS, while probabilistic routines are all programmed in the computer algebra program MAPLE 2019. Additionally, one may employ MAPLE to derive relative entropies formulas based on the first two moments or distribution parameters for the non-Gaussian variables. It should be underlined that the response functions are assumed to have polynomial form of the given input uncertain parameter, whose order and coefficients are found using the Least Squares Method fitting procedures. Computer analysis would concern both some classical linear elasticity engineering problem and inelastic incremental analysis. It will be parametrized with the input coefficient of variation of the given uncertainty sources to check possible availability limits.

Acknowledgment

This work has been prepared in the framework of the research grant OPUS no 2021/41/B/ST8/02432 entitled 'Probabilistic entropy in engineering computations' and sponsored by The National Science Center in Poland.

WAVE ABSORBING BOUNDARIES IN DYNAMIC ANALYSES OF WIND TURBINES

Marlene Steggewentz[1], Otfried Beilke[2], Michael Stahl[1]

[1] ICG Ingenieure GmbH, Germany
[2] Civil Engineering, Jade University of Applied Sciences, Germany

msteggewentz@icg-ing.de

Soil is an infinite domain. A numerical model, however, is spatially limited. When the system is subjected to dynamic loads, the emitted waves can't propagate into infinity and reflections occur at the model's artificial boundaries. These reflected waves overlap with the initial excitation and distort the results strongly. Extremely idealized boundaries hardly deliver reliable results for dynamic soil-structure-interaction problems.
Various approaches to prevent wave reflections exist in the literature. The type of boundaries has a great influence on the resulting deformations and stresses and consequently affect the calculation of stability and usability. The approaches often involve high modelling effort or computing time. Therefore, the selection of appropriate boundaries requires a thorough examination, especially in dynamic analyses.
This paper's aim is to perform a dynamic analysis of a wind turbine using different methods as absorbing boundaries in ANSYS: spring damper elements, infinite elements, and perfectly matched layers (PML). The specific aspects and the application of these methods are examined and compared.
Assuming that the soil behaves linear elastic in dynamic analyses the soil's stiffness can be simplified as a spring and the damping characteristic as a damper element. This system has a very low number of degrees of freedom and consequently very low computing time.
Based on the spring-damper-system, Lysmer and Kuhlemeyer (1969) developed viscous boundaries that absorb the energy of incoming waves. These viscous boundaries can be applied as infinite elements.
PML, a lossy material that absorbs the wave's energy, were originally developed by Berenger (1993) for an application in electromagnetism and were refined for various other fields, like soil dynamics.
As the results show, all three methods are suitable to simulate the dynamic soil-structure-interaction of a wind turbine without causing reflections at the boundaries. However, each method entails different advantages and disadvantages. Using spring-damper-elements a thorough study of an appropriate approach to calculate the spring-damper-coefficients is required. Layered half-space can be problematic. Infinite elements and PML can be applied without any general limitations of the half-space composition. A finite-infinite-element-model requires an interface domain between the near and far field that is subjected to specific requirements. The application of PML is not subjected to any numerical or mechanical requirements. However, the application is limited to harmonic analyses and three-dimensional models. This paper presents a general description of the different methods and a suitable procedure for choosing an adequate model.

FRACTURE ENERGY OF WOODEN MEMBERS UNDER CONTACT EXPLOSION

Artur Szlachta[1], Piotr Sielicki[1], Tomasz Grajewski[1]

[1] Poznan University of Technology, Poland

aarczii89@gmail.com

Wood is a basic construction material in military operations. One of the skills exercised while training sappers ins the dynamic destruction of wooden structures. The military instructions available refer too generally to the issue of wood strength. In our preliminary studies, Iit turneds out that taking into account even a few variables, such as diameter, element moisture, grade, it is insufficient to precisely design the charge, which would cause a wooden element destruction. Therefore, the compression tests of birch, oak and pine in three directions were performed to define orthotropic constitutive models, the results were used as the output data of the numerical models. Analyses of the finite element method of the destroyed logs with the use of contact cylindrical charges for selected wood species were performed and the destructive values for various diameters were calculated. It was found that the charge masses obtained from military instructions are overestimated comparing to the numerical outcome. I would consequence in using more explosives than needed during combat. The photographs from the preliminary tests of destruction of single wooden elements separated from the structure are presented below.

DISCRETE WAVELET TRANSFORM APPLICATION FOR DETECTING DAMAGE IN BARS AND NODES OF TRUSS STRUCTURES

Anna Knitter-Piątkowska[1], **Olga Kawa**[1]

[1] Institute of Structural Analysis, Poznan University of Technology, Poland

anna.knitter-piatkowska@put.poznan.pl

An inherent part of life cycle and operational time of the structure is its wear and tear and the possibility of defect occurrence. The issue of early detection, location and estimation of structural damage is one of the most important engineering problems, since it is strictly related to the safety and durability of the facility. For many years, numerous scientists have developed methods of identifying damage with an emphasis on non-destructive testing. A great deal of these techniques rely on the structural response signals. A very promising tool for signal analysis, which has been the subject of intensive studies in recent years, is Wavelet Transform (WT), also in its discrete form (DWT). The DWT has been proven to be a very effective tool for defect detecting in the structures such as beams or plates [1] subjected to static or dynamic load. In the current research the authors focus on detection and localization of the deterioration in the bars and nodes of the truss structure. The system of flat truss girders subjected to classic code loads is examined. The damage is modeled as stiffness reduction along a very small portion of the selected bars (lower, upper chord or the diagonals) length or the detachment of a truss bar at a small distance in the truss node. The structural static response signal, e.g. displacements, is derived while employing the Finite Element Method (FEM) in terms of beam or surface finite elements and computational programs (Robot Structural Analysis, AxisVM).

The response signal is then analyzed and transformed while applying DWT procedures. The multiresolution signal analysis with the use of Mallat pyramid algorithm [2] is engaged. The set of Daubechies and Coiflet families of wavelet functions is applied to the preliminary analysis. An evident disturbances of the transformed signal in the transformation window are expected as the indicator of the presence of the defect.

References

[1] Knitter-Piątkowska, A., Guminiak, M., Application of 1-D and 2-D Discrete Wavelet Transform to crack identification in statically and dynamically loaded plates, Engineering Transactions, 68(2), pp.137-157, 2020.

[2] Mallat, S.G., A theory for multiresolution signal decomposition: The wavelet representation, IEE Transactions on Pattern Analysis and Machine Intelligence, 11, 7, pp. 674-693, 1998.

EXPERIMENTAL AND NUMERICAL INVESTIGATION OF MATERIAL AND DAMAGE BEHAVIOUR OF 3D PRINTED POLYAMIDE 12

Daniela Schob[1], Robert Roszak[1], Krzysztof Kotecki[2], Dariusz Kurpisz[2], Matthias Ziegenhorn[1]

[1] Brandenburg University of Technology Cottbus-Senftenberg, Germany
[2] Mechanics, PTU Poznan, Poland

schobdan@b-tu.de

In previous work, the material model of Chaboche and the damage model of Gurson-Tvergaard-Needleman were successfully applied to additively manufactured polyamide 12 under quasi-static and cyclic loading. In the present work, the model was adapted for shear loading. The characterization challenge was to account for the inhomogeneous material structure. An initial porosity of 5% was determined by means of X-ray and CT examinations. The Gurson-Tvergaard-Needleman model takes into account pores and their growth, nucleation and coalescence, but not cross-linking under shear stress. For the implementation, the existing model was modified with the approach of Reis. Along with this, the application of commercial FE software and the existing material libraries is limited. For this reason, the modified model was implemented in user material subroutine (UMAT). The experimental results of digital image correlation during the shear test were used to validate the simulation. The experimentally determined deformation field was compared with the simulation results. An excellent correlation between experiment and simulation could be achieved for the shear loading of PA12.

MECHANICAL RESPONSE OF PRE-BUCKLED HONEYCOMB STRUCTURES MANUFACTURED ADDITIVELY

Marcin Sarzyński[1], **Paweł Płatek**[1], **Paweł Baranowski**[2], **Igor Czernichowski**[1]

[1] Faculty of Mechatronics, Armament and Aerospace, Military University of Technology, Poland
[2] Faculty of Mechanical Engineering, Military University of Technology, Poland

marcin.sarzynski@wat.edu.pl

Additive Manufacturing (AM) is one of the most disruptive technique that attract attention of many researchers. Offered design freedom, possibility of fabrication objects with complex shape cause that AM techniques are widely implemented in design process in many branches of industry. Based on the literature study it can be found that AM technique are used in development of new material structures that depending on the applied topology demonstrate a low mass and specific mechanical properties like high mechanical stiffness or high energy absorption capacity. This kind of material structures (e.g. honeycomb) indicate an anisotropic mechanical properties that results from the geometrical stiffness and compression loading direction. Taking into consideration the potential range of application this types of structure it can be observed that it is generally limited to one of following requirements: high stiffness or high energy absorption capacity. The main aim of the paper is related to experimental and numerical studies of honeycomb based material structures with additional pre-buckling initiators introduced on the side surfaces to fulfill both mentioned above engineering requirements. Proposed investigations assumed to design and evaluation of new material structure topologies that allow obtaining a similar mechanical response (loading force - deformation curves) under in-plane and out-of-plane compression loading scenarios. The main problem for designing the geometry of structures was solved by definition of the triangle waveform parameters (amplitude and period) ensuring a constant level of the stress plateau for in-plane and out-of-plane loading. The designed structures were examined experimentally and using the finite element analysis. The material structure specimens were manufactured using the Additive Manufacturing Fused Filament Fabrication (FFF) technique. The PLA-modified polymer material was used in the fabrication process. The comparison of the results of experimental tests and numerical simulations made it possible to verify the initial-boundary assumptions adopted in the numerical model. Based on the obtained results it is possible to conduct an further optimization analysis aimed to define the most effective shape of the orthotropic honeycomb topology that ensure the high energy absorption capacity.

IDENTIFICATION OF ELASTIC MATERIAL PARAMETERS BASED ON DIGITAL IMAGE CORRELATION RESULTS

Marcin Nowak[1], **Paweł Szeptyński**[2], **Michał Maj**[1]

[1] Mechanics of Materials, Institute of Fundamental Technological Research, Polish Academy of Sciences, Poland
[2] Division of Structural Mechanics and Material Mechanics, Faculty of Civil Engineering, Cracow University of Technology, Poland

nowakm@ippt.pan.pl

The goal of the work is to present the new methodology of determination of elastic constants in constitutive relations that can be applied to any elastic material. The approach is based on the Digital Image Correlation (DIC 2D) method in combination with stress field determination. It is worth emphasizing that there are no special requirements regarding the sample geometry, as well as the boundary conditions used to enforce the deformation process. Only the experimentally measured displacement field and the external force measured by the testing machine are needed for numerical calculations.

Parameters of the assumed linear constitutive law are found as an optimal set of values minimizing the measure of imbalance between external load and determined internal forces corresponding with assumed imaginary cut-surfaces. The imbalance function is assumed as a weighted sum of squares of residues of equilibrium equations, each written down for a finite-size part of a sample cut through with each particular surface. Optimization task is solved with the use of the Nelder-Mead downhill simplex algorithm.

The efficiency of the algorithm is verified with two types of materials: 310S steel and CFRP tape. Experimental studies are carried out on two types of samples with imperfections in the form of either eccentrically placed circular hole or undercut on one side. Each of the samples is stretched until the limit of elastic range is exceeded. The introduced imperfections cause that the deformation distribution is heterogeneous from the very beginning of the process. During deformation process the sequence of images is recorded using pco.edge 5.5 camera working in visible range. On the basis of the obtained image sequences, the displacement field is determined using ThermoCorr 2D digital image correlation program.

Additionally, the correctness of the proposed method is verified using numerically generated displacement field from FE simulations as experimental data. This approach allows to mimic the real experiment and estimate errors. Preliminary calculations show that the proposed method can be successfully used for identification of constitutive parameters for various types of materials.

THE DUAL-PHASE LA2ZR2O7 + 8YSZ ATMOSPHERIC PLASMA SPRAYED THERMAL BARRIER COATINGS: DETERMINATION OF THERMAL CONDUCTIVITY AND THERMAL STRESS DISTRIBUTION

Anna Jasik[1]

[1] Department of Material Technologies, Silesian University of Technology, Poland

anna.jasik@polsl.pl

The article presents problems related to the new concept of thermal barriers coatings (TBC) based on dual-phase systems of the Ln2Zr2O7 + 8YSZ type. Conventional TBC's are systems consisting of three elements: the base material - usually a nickel-based heat-resistant alloys, an intermediate layer designed to provide adequate oxidation resistance, and a ceramic outer layer with insulating properties. The usually used and dominant material in insulating layers is zirconium oxide modified with yttrium oxide (ZrO2 x 8% wt Y2O3 - 8YSZ) deposited by plasma spraying methods or directly from the gas phase. Its main limitation is the tendency to phase transformation under the influence of temperature and in the assist of liquid salt or glass deposits.

An alternative group of materials is zirconates of rare earth elements of the Ln2Zr2O7 with a pyrochlore type lattice. They are characterized by much higher lattice imperfections and thus a lower thermal conductivity coefficient value than 8YSZ. The disadvantage of TBC coatings based on Ln2Zr2O7 compounds is their high tendency to crack and spall, resulting from the higher value of Young's modulus than conventional coatings. The solution to this problem is using dual-phase and two-layer coatings systems consisting of an external zirconate layer and an inner 8YSZ layer, and such internal morphology provides better tolerance to deformation than mono-phase coatings. Thermal conductivity studies have shown that the insulation properties of this type of TBC coatings are even better than for a coating based only on rare earth zirconates. This phenomenon suggests that the factor strongly influencing the decreasing of thermal conductivity is the interphase boundaries and the so-called Kapitza resistance.

Therefore, the concept of composite (dual-phase) coatings of the Ln2Zr2O7 + 8YSZ type, obtained from a mixture of both types of powders in equal mass proportions, was proposed. As a result, composite ceramic coating structures with high interphases splat-to-spalt boundaries density were produced and characterized by the lowest thermal conductivity and high strain tolerance. These assumptions were verified under the conditions of long-term heating (2000 h) at the temperature of 1100 ℃, during which no cracking and peeling effects were found, as well as under the thermal conductivity measurement. The thermal stresses in the Ln2Zr2O7+ 8YSZ coating were also determined.

The author would like to acknowledge the support provided by the Silesian University of Technology, Gliwice, Poland, under the Faculty of Materials Engineering, Statutory Found for 2022 y. no 11/030/BK_22/1082.

COMBINED EXPERIMENTAL AND ANALYTICAL APPROACH TO DETERMINE LIMIT LOAD OF A SLENDER REINFORCED CONCRETE COLUMN IN OPERATIONAL CONDITIONS

Milena Drozdowska[1], Agnieszka Tomaszewska[1], Mateusz Sondej[1]

[1] Department of Structural Mechanics, Gdansk University of Technology, Poland

mildrozd@pg.edu.pl

Steel roof lattice girders simply supported on slender reinforced concrete (RC) columns is a typical structural concept wildly used for large production and warehouse facilities. Optimal design of this structures is concentrated on minimizing cross section of the columns. The slender columns are highly sensitive for wind forces and second order effects which can dramatically reduce column capacity. Mechanical behaviour of reinforced concrete is complex - steel reinforcement with relatively simple elastic properties is combined with concrete defined by non-linear stress-strain curve, creep, cracking stresses and tension stiffening effect. The main difficulties lay in determining second order effects which depends also on boundary conditions, column height and number of columns carrying vertical loads in analysed plane.
The paper relates the problem of limit load determination of a RC column, with respect to boundary condition determined in in situ measurements. A case of 12 m high column with 0.5 x 0.45 m cross-section is considered, which was designed as fixed in the ground. The column represents one of many in an extensive construction of industrial facility. To determine the support condition one column not yet connected to other elements was subjected to vibration measurements in operational conditions. Natural frequencies and mode shapes of the object were identified using Eigensystem Realization Algorithm (ERA). Three first modal pairs have been identified for two orthogonal directions of the column. The obtained modal results showed that the static diagram of the column is inconsistent with the assumed cantilever diagram - the column is supported in an elastic manner. Further identification allowed to determine rotational spring stiffness which is instead of a full rotational fixation.
Next, stability of the construction was checked according to Eurocode 2 general method procedure, including the identified elastic support in second order nonlinear analysis. The influence of boundary condition on column stability was analysed in detail. The results for three Eurocode column design procedures: the general method, the nominal stiffness method and the nominal curvature method, were compared. It was shown that operational modal analysis can be successfully used to identify parameters for boundary conditions and the obtained data can enhance stability analysis of the structure.

FEM SIMULATION AND OPTIMIZATION OF FINE-BLANKING PROCESS FOR ALUMINUM ALLOY SHEET

Łukasz Bohdal[1], Leon Kukiełka[1], Radosław Patyk [1], Mateusz Miksza[1], Katarzyna Kośka [1]

[1] Mechanical Engineering, Koszalin University of Technology , Poland

lukasz.bohdal@tu.koszalin.pl

The primary objective of the fine-blanking process is to reduce the fracture zone depth, rollover zone and burr height on workpiece cut surface. In the case of aluminum, according to the material properties, it is possible to speak of very plastic materials which are very difficult to cause a sudden fracture. If it occurs, the quality of the cut surface is low and it is characterized by high fracture roughness and burr formation. A significant problem of this type of machining is the frequent appearance of undesirable random defects in the cross-section of the cut surface workpiece and on the cutting edge of the tool. As a result, this leads to an increase in temperature in the contact zone of the tool with the sheet, contributing to faster wear of the cutting tools, and an increase in waste and energy consumption in the process. Therefore, it is necessary to explain the causes of these defects and to develop ways to avoid them. The paper presents the results of numerical simulations with the use of FEA and experimental research aimed at analyzing the influence of the main parameters of the process on the formation and quality of the cut edge for selected aluminum alloy. The influence of the punch rounding radius on the fatigue wear was investigated. In experimental studies, the abrasive wear process was primarily investigated. An optimization process was carried out to determine the process conditions that would ensure the highest quality of the cut edge and the greatest durability of the cutting tools.

EXPERIMENTAL-NUMERICAL METHODS FOR STUDYING SHOCK-INDUCED FRACTURE OF ROCK

Paweł Baranowski[1], **Michał Kucewicz**[1], **Jerzy Małachowski**[1], **Mateusz Pytlik**[2]

[1] Institute of Mechanics & Computational Engineering, Military University of Technology, Poland
[2] Conformity Assessment Body, Central Mining Institute, Poland

pawel.baranowski@wat.edu.pl

New methodology to study rock brittle failure using two original small-scale blast setups is presented. Experimental tests were conducted using cylindrical and disc specimens of dolomite material, both with a borehole in their center. A detonation cord and a blasting cap were fitted inside the borehole to induce cracking and fracturing of the specimens. The laboratory investigations were simulated using the Johnson-Holmquist II (JH-2) material model and the failure of the dolomite was compared with the test observations. Comparisons of qualitative and quantitative results confirmed that the developed models with the JH-2 constitutive model captured very well the actual testing data. The main purpose of the presented studies is to present new methods for studying rock behavior under blast loading and for validating the numerical and constitutive models used for rock simulations.

DYNAMIC CONTACT ANALYSIS OF A ROTATING ANNULUS USING REDUCED ORDER SYSTEM APPROACH

Akilesh G[1], **Manoj Pandey**[1]

[1] Mechanical Engineering, Indian Institute of Technology, Madras, India

me19s074@smail.iitm.ac.in

Design problems using high fidelity numerical methods such as Finite Element Analysis (FEA) can be computationally intensive, especially if they require multiple runs for different loading conditions or varying parameters. Hence reduced-order models (ROM) which can reproduce the simulation results with high accuracy, while working at a very low computational budget are desirable. Subspace projection-based ROMs are widely used for the analysis of linear systems, using linear eigenmodes as projection basis and can be extended to nonlinear systems using empirical eigenmodes obtained from Proper Orthogonal decomposition (POD). Here the solution is assumed to be lying in a subspace whose size is much smaller than the full system.
Problems involving moving contact are difficult to handle for such a procedure due to moving boundary conditions of the underlying PDE. Moreover, to achieve significant speedups, we need to compute the approximation of tangent matrices and nonlinear force vectors instead of exact quantities before projecting those onto the subspace. This process is often referred to as hyper reduction in literature. Here we use the hyper reduction approach proposed by [1] called the energy-conserving sampling and weighting (ECSW) towards reduced-order dynamic analysis of Hyperplastic annulus rolling, incorporating the geometric and material nonlinearities in addition to contact. To further reduce the ROM size, instead of approximating the solution using the global basis, we approximate the solution in multiple lower-dimension subspaces called local bases [2]. During the online solution phase of the problem, a local basis is chosen such that it captures the current high dimensional solution. This approach can be used along with hyper reduction to further improve the computational speed.
The simulations are performed using commercial Finite Element software Abaqus and the ROM based results are simulated and verified using MATLAB. An annulus modelled with compressible hyper elastic Neo-Hookean material model is first brought into contact with the frictionless rigid surface by a static force loading step to mimic a tyre being loaded statically and its results are used as the initial condition to the dynamic step. The contact constraints are enforced using the penalty method. In the dynamic step, the inner surface of the annulus is subjected to prescribed constant angular velocity. The time integration scheme used was Hilber-Hughes-Taylor (HHT) method. Due to rotation, the active contact zone keeps changing at every time increment. The ROM was first created without hyper reduction but with local bases. The ROM size using 10 local bases was around 200 (Full dimensional model size was 8112) for the dynamic step and it had very good accuracy. Applying hyper reduction gave good results and further speedup to the static force loaded analysis.
In conclusion, the results show a very good match for displacement and contact forces with a model that is orders of magnitude smaller than the full order system. In future we aim to use hyper reduced ROM and extend the procedure to tyre rolling analysis.

References

[1] Chapman, T. A., Farhat, C., Alonso, J. J., Kochenderfer, M. J., & Stanford University. (2019). Nonlinear model order reduction for structural systems with contact.
[2] Farhat, C., Chapman, T. and Avery, P. (2015), Structure-preserving, stability, and accuracy properties of the energy-conserving sampling and weighting method for the hyper reduction of nonlinear finite element dynamic models. Int. J. Numer. Meth. Engng, 102: 1077- 1110. doi: 10.1002/nme.4820.