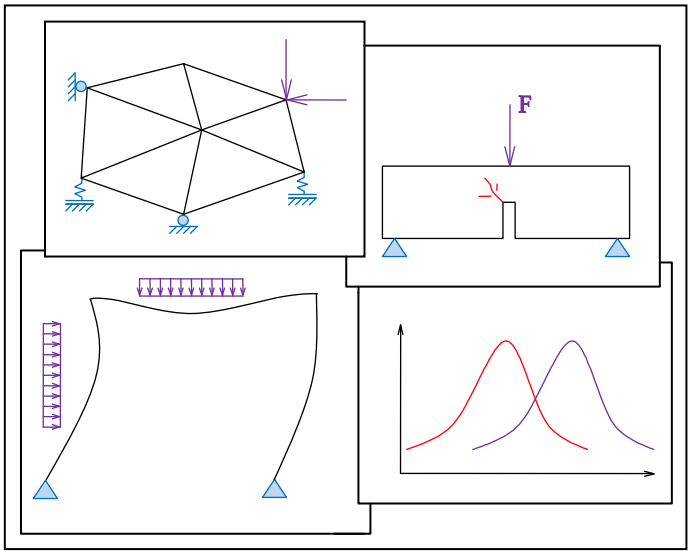
VSB TECHNICAL UNIVERSITY OF OSTRAVA | FACULTY OF CIVIL ENGINEERING

19. ročník mezinárodní konference

Modelování v mechanice 2021

21. - 22. 10. 2021

Sborník rozšířených abstraktů



19th International Conference

Modelling in Mechanics

21st and 22nd October 2021

Proceedings of extended abstracts

ISBN 978-80-248-4560-9 (Print)
ISBN 978-80-248-4561-6 (Online)

TABLE OF CONTENTS / OBSAH

NÁVRH OCELOVÉ KONSTRUKCE PRO EXPERIMENTÁLNÍ TESTOVÁNÍ ŽELEZOBETONOVÝCH PŘÍHRADOVÝCH VAZNÍKŮ

Pavel DOBEŠ[1,2], *Petr MYNARČÍK*[1], *David MIKOLÁŠEK*[2], *Vít KŘIVÝ*[2]

[1]Experimentální a diagnostické stavební centrum, Fakulta stavební, Vysoká škola báňská – Technická univerzita Ostrava

[2]Katedra konstrukcí, Fakulta stavební, Vysoká škola báňská – Technická univerzita Ostrava

pavel.dobes1@vsb.cz, petr.mynarcik@vsb.cz, david.mikolasek@vsb.cz, vit.krivy@vsb.cz

Klíčová slova

Ocelová konstrukce, předpjatý beton, diagnostika stavebních konstrukcí, železobetonový příhradový vazník, experimentální testování, numerické modelování.

1. Úvod

Příspěvek je zaměřen na aktuálně rezonující problematiku spínaných železobetonových příhradových vazníků typu SPP6-18/6, které byly vyvinuty na přelomu šedesátých a sedmdesátých let minulého století státním podnikem Závody inžinierskej a priemyselnej prefabrikácie, š.p. Prefabrikovaný konstrukční systém je v současnosti centrem pozornosti z důvodu několika evidovaných kolapsů tohoto příhradového vazníku v uplynulých letech [1]. Vzhledem k aktivní angažovanosti v problematice v oblasti stavebně technického průzkumu se podařilo pracovišti EXC Stavební fakulty VŠB-TUO získat pro podrobnou analýzu několik nepoškozených exemplářů vazníku SPP6-18/6. V tomto příspěvku je popsána numerická analýza ocelové konstrukce, která bude využita pro zatěžovací zkoušku problematického konstrukčního prvku.

2. Numerická analýza ocelové konstrukce

Pro potřeby zatěžovací zkoušky je navržen prostorový ocelový exoskelet, do kterého se umístí zkoušený ŽB vazník. Obě konstrukce tvoří uzavřený statický systém, který je při vnesených silách v rámci experimentu ve vzájemné statické rovnováze. Ocelová konstrukce se skládá ze dvou hlavních rámů. Pásové pruty rámů jsou rozděleny na přímé segmenty a spolu se svislicemi jsou spojovány v T, respektive L styčnících pomocí čelních desek a šroubů. Ztužení polí rámů je realizováno pomocí táhel z kulatiny. Rozpěrné příčníky mezi rámy slouží k uložení vazníku (krajní) a vnášení zatížení (mezilehlé) při zatěžování. Pro výrobu ocelového exoskeletu je navržena konstrukční ocel S355J2.

Návrh prvků vychází z globální analýzy v programu SCIA Engineer (GMNIA). Pro ocelové prutové prvky je použit ideálně pružno-plastický materiálový model se zpevněním o hodnotě E/10000. Na základě platných norem jsou rovněž stanoveny lokální a globální imperfekce [2]. Pro model ŽB vazníku jsou použity skořepinové prvky a Drucker-Pragerův pružno-plastický materiálový model. Nelineární výpočet aplikuje Newton-Raphsonovu přírustkově-iterační metodu a řešič dle Lanczose pro stabilitní analýzu. Následný návrh a posudek spojů je proveden pomocí programu IDEA StatiCa.

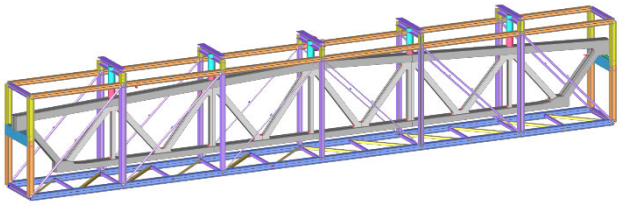


Fig. 1: Numerický model ocelové konstrukce a ŽB vazníku.

3. Závěr

Cílem tohoto příspěvku je představit návrh ocelového exoskeletu pro potřeby experimentálního testování železobetonových příhradových vazníků v prostorách EXC na Stavební fakultě VŠB-TUO.

Reference

[1] ŠPALEK, R. VÝZVA: Opakované havárie betonových předpínaných vazníků. *Zprávy a informace ČKAIT [online]*, 2019.

[2] Normy řady ČSN EN 1993-1

DESIGN OF MEMBRANE STRUCTURE AS A TEMPORARY ROOFING SYSTEM

Nela FREIHERROVA[1], *David JURAČKA*[1], *Marek KAWULOK*[1], *Radim ČAJKA*[1], *Martin KREJSA*[1]

[1]Department of Structural Mechanics, Faculty of Civil Engineering, VSB-Technical University of Ostrava, Ludvika Podeste 1875/17, 708 00 Ostrava-Poruba, Czech Republic

nela.freiherrova@vsb.cz, david.juracka.st@vsb.cz, marek.kawulok.st@vsb.cz, radim.cajka@vsb.cz, martin.krejsa@vsb.cz

Abstract. *This contribution is focused on the design analysis of membrane structure. The membrane structure is designed for roofing of the experimental stand, which is in the campus of Faculty of Civil Engineering, VSB – Technical University of Ostrava and is currently not protected against unfavorable weather conditions. The aim of this paper is to give an overview of the design of the temporary membrane structure starting from the architectural visualization and continuing to deal with structural and dispositional parts of the design using the RFEM software.*

Keywords

Membrane structures, tensile structures, lightweight structures, roofing system, form finding, numerical modelling, nonlinear analysis, RFEM software, Rhino software.

1. Introduction

Membrane structures combine surface material and supporting elements, mostly made of steel. When designing such a structure, the first step is to find the optimal shape [1]. This process is called Form – Finding. and deals with finding the optimal deflection and visual shape based on the given boundary conditions. It is necessary to apply pre-stressing into the structure which prevents inversion of the curvature and after that, the structure can be loaded. The initial shape of membrane structure must be designed in such a way that only the tensile forces can occur in the structure.

2. Design of membrane roofing

It was necessary to consider the type of surrounding buildings, obstacles, and the slope of the terrain.

After determining the boundary conditions, a suitable shape of the membrane structure was solved. The final shape can be seen in Fig. 1. The new prestressed shape is then used as the initial state for the structural analysis.

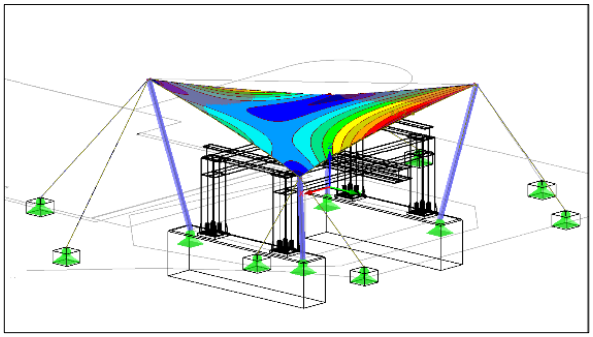


Fig. 1: Final shape of membrane structure

3. Conclusion

The aim of this paper was to give an overview of the design of the temporary membrane structure starting from the architectural visualization and continuing to deal structural and dispositional parts of the design.

Acknowledgements

The financial supports of the grant program financed by Ministry of Education, Youth and Sports of the Czech Republic through VSB – TU Ostrava (SGS SP2021/80) is highly acknowledged.

References

[1] NĚMEC, I. and R. LANG. Design of initial shape of membrane structures. Transactions of the VSB - Technical University of Ostrava, Civil Engineering Series. 2013, 13 (2), p. 105-109. ISSN: 1213- 1962

FRACTURE CHARACTERISTICS OF FIBER REINFORCED SLWAC FROM 3PBT USING INVERSE ANALYSIS

Marie HORNAKOVA[1], *Petr LEHNER*[1], *Jacek KATZER*[2]

[1] Department of Structural Mechanics, Faculty of Civil Engineering, VSB-Technical University of Ostrava, Ludvíka Podéště 1875/17, 708 33 Ostrava-Poruba, Czech Republic

[2] Civil Engineering Center, Faculty of Geodesy, Geospatial and Civil Engineering, University of Warmia and Mazury, ul. Prawochenskiego 15, 10-720 Olsztyn, Poland

marie.hornakova@vsb.cz, petr.lehner@vsb.cz, jacek.katzer@uwm.edu.pl

Abstract. This paper describes the implementation of inverse analysis in the case of experimentally obtained results. The method is presented on structural lightweight concrete (SLWAC) made of fine red ceramic waste aggregate, expanded clay coarse aggregate, and reinforced by steel fiber in 1.5% of concrete volume. The material characteristics, especially data from the three-point bending test (3PBT), have been used for the development of the material model allowing further numerical modelling of fiber-reinforced SLWAC.

Keywords

Waste red ceramic, copper coated crimped steel fiber, fracture characteristics, inverse analysis.

1. Introduction

The randomly distributed fibers are added into concrete to enhance the post-cracking residual tensile strength and ductility. In this project, the fiber is in crimped shape made of copper-coated steel (CCSF), fine aggregate is waste material from a low-quality red ceramic airbrick and commercially available expanded clay aggregate is utilized as a coarse aggregate. The process of designing and results of testing the mechanical and durability characteristics of the FR SLWAC were described in detail in previous papers. The fiber together with using of waste material in a creation of a concrete mixture represent a promising future in the field of civil constructions.

The aim of the study is to present a material model of FR SLWAC1.5% defined by the obtained material characteristics and experimentally measured data of load and crack mouth opening displacement (CMOD) from the 3PBT conducted in accordance with European Standard. The 2D model (Fig. 1) was created in software GiD + ATENA Studio using the finite element method (FEM) and fracture-plastic constitutive material model for nonlinear analysis of concrete - CC3DNonLinCementitious2User. The tensile function of the material model has been modified until matching the experimental data.

2. Results and Conclusions

The designed fiber-reinforced SLWAC1.5% has been tested by 3PBT and its fracture parameters were defined by inverse analysis. The data from the experiment and the results from the numerical simulation are shown in Fig. 2. Good compliance between the experimental results and numerical simulation of load-CMOD curves was obtained.

Fig. 1: 2D numerical model and boundary conditions in GiD.

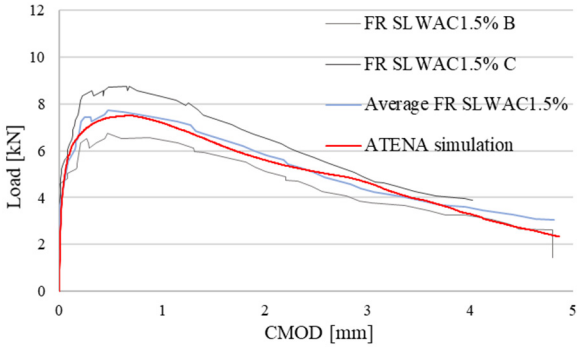


Fig. 2: Results from the 3PBT of SLWAC1.5% and from numerical simulation in ATENA.

Acknowledgement

The paper has been elaborated in the framework of the grant programme financed by Ministry of Education, Youth and Sports of the Czech Republic through VSB – TU Ostrava (SGS SP2020/81) and grant programme „Support for Science and Research in the Moravia-Silesia Region 2018" (RRC/10/2018).

Numerical Homogenization of Mechanical Metamaterials with 3D Structure

Nazar HUDYMA, Jan PODROUŽEK

Institute of Computer Aided Engineering and Computer Science, Faculty of Civil Engineering, Brno University of Technology,
Veveří 331/95, 602 00 Brno, Czech Republic

Nazar.Hudyma@vutbr.cz, podrouzek.j@fce.vutbr.cz

Abstract. *Mechanical metamaterials are becoming more common in last years due to reduced difficulty in the fabrication of complex 3D structures (on macro/meso/micro scale) by application of aditive manufacturing. To design structures on this level of complexity and detail, the computer aided methods are needed. In particular solving mechanical properties of metamaterials with basic FEM software is inefective to the extent that heterogenical metamaterial has to be simplified as anisotropic homogenical material. Basicly the proces of numerical homogenization predicts behaviour of metamaterial at its structural boundaries. For instance the structure on meso scale level is transformed to macro scale level homogenous material. In this paper is shown actual state of the art in the field of numerical homogenization.*

Keywords

Aditive manufacturing, numerical homogenization, metamaterial.

1. Introduction

In engineering there has been allways motivation to use as little material as possible to be most effective at different aspects as cost, weight, stifness, etc.. This idea was transformed into mechanical metamaterials with other mechanical properties than the original materials had. For many years it has been unimaginable to manufacture materials with these properties, but with development in field of aditive manufacturing (AM) it became possible. Superstructures with complex substructures (metamaterials) are way more time consuming to design and compute, but are way more cheaper to produce than a part with simpler geometry and homogenous material (complexity paradox).

Metamaterial fabricated by AM may also vary on different paths of printing and layering. Thus design of structure must have printing settings included in numerical analysis. The material properties may vary on every scale and direction. The homogenization is used in order to obtain effective material properties for superstructure. Homogenization transforms well detailed and complex structure with voids or combination of materials with "simpler" solid material.

Homogenization requires significant computational resources and well known micro and meso scale of fabricated metamaterial. The problem can be solved by constructing implicit geometry of material (meso scale) and experimentaly measururing material properties on the end of printing medium (micro scale). The results of homogenization are tensors of various ranks (as vector, matrix, etc.).

2. Conclusion

In future there will be bigger amount of structures fabricated by AM. With free complexity provided by AM there will be more common use of optimization tools in design of superstructures on all scales of structure. The homogenization saves computing time reducing load on FEM solver and user by simplifying computational model. The difference becomes substantional with complex periodical patterns as gyroids or teseract and other strut based patterns.

Acknowledgment

This contribution has been completed thanks to the financial support provided by Brno University of Technology under project No. FAST-J-21-7512

ANALYSIS OF FORCED VIBRATION DAMPING WITH THE USE OF HYPERPLASTIC MATERIALS

Anna JASKOT, Maciej MAJOR

Department of Civil Engineering, Faculty of Civil Engineering, Czestochowa University of Technology, ul. Akademicka 3, 42-201 Czestochowa, Poland

anna.jaskot@pcz.pl, maciej.major@pcz.pl

Abstract. *A solution of forced vibration damping through the use of hyperelastic materials have been proposed in the paper. Most structures can be easily simulated on the basis of a linear elastic material model. However, when linear elasticity ceases to be sufficient, then nonlinear models are adopted, e.g. hyperelastic materials. The aim of this study was to investigate the behavior of the structure under the influence of external forces with the use of materials showing high fatigue resistance to compression with simultaneous consideration of vibration damping. The model presented in the paper has been subjected to an exciting force of the same value in both considered cases. The paper presents a dynamic analysis using the finite element method in the ANSYS program. The comparison concerned the use of hyperelastic materials with the use of the deformation energy model of hyperelastic incompressible materials according to the Mooney-Rivlin model.*

Keywords

Forced vibrations, damper, Mooney-Rivlin model.

1. Introduction

Mechanical vibrations are undesirable in most cases. Due to the fact that in many cases it is impossible to remove the source of vibrations, various methods of damping and minimizing their impact are used. A solution for damping vibrations through the use of rubber-based materials, which are defined as incompressible materials, have been propose in the paper. In the case of modeling linearly elastic materials according to the FEM theory, it is enough to define two material constants (Young's modulus and Poisson's number). Hyperelastic materials are able to undergo large elastic deformations under the influence of forces but exhibit non-linear behavior and their deformation is not directly proportional to the applied load [1]. In the case of hyperelastic material models, additional material parameters must be defined. The material characteristics of typical hyperelastic materials have been shown in Fig. 1.

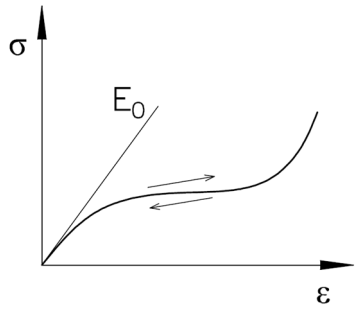


Fig. 1: Stress-strain characteristic of typical hyperelastic materials [2].

Based on this dependence and taking into account the deformation range in the simulation, a model of the deformation energy of hyperelastic incompressible materials has been selected. In the deformation range from 10% to 300% deformation, the best approximation is provided by the Mooney-Rivlin model, the form of which is presented in the general formula:

$$W = C_{10}(I_1-3)+C_{01}(I_2-3), \quad (1)$$

where: I_1 and I_2 – invariants describing the strain energy density, C_{10} and C_{01} – material constants.

The results of numerical tests performed in the ANSYS/Mechanical program, as well as the dynamic interactions, and a proposition of vibration damping solution has been presented in this work. The model can be the basis for further numerical and experimental research.

References

[1] JASZAK, P. Modelowanie gumy za pomocą metody elementów skończonych, Elastomery. T. 20, no.3. 2016, pp. 31-39 (in polish).

[2] MAJOR, M., RÓŻYCKA, J. Gumopochodne materiały hipersprężyste – omówienie i kryteria praktycznego zastosowania. Budownictwo. T. 17, 2011, pp. 134-145 (in polish).

EXPERIMENTÁLNE TESTOVANIE DREVENÉHO RÁMOVÉHO SPOJENIA

Marek JOHANIDES, Antonín LOKAJ

Katedra konstrukcí, Fakulta stavební, VŠB – Technická univerzita Ostrava,
708 00 Ostrava - Poruba, Česká republika

marek.johanides@vsb.cz, antonin.lokaj@vsb.cz

Abstrakt. *Príspevok sa zaoberá inováciou dreveného rámového spojenia vytvoreného zo systému dvoch stojok a priečle. Toto rámové spojenie bolo vytvorené z lepeného lamelového dreva triedy GL24h. Pre spojenie boli namiesto štandardne využívaných mechanických spájacích prostriedkov, kombinácie svorníkov a kolíkov, použité celozávitové skrutky triedy pevnosti 10.9. Celkovo boli otestované tri variácie spojenia. V prvom teste neboli jednotlivé komponenty rámu nijako spevnené. V druhom teste bola pomocou dvoch celozávitových skrutiek spevnená priečla a v treťom teste boli pomocou dvoch celozávitových skrutiek spevnené stojky aj priečla. V príspevku boli experimentálnym testovaním zistené maximálne únosnosti a rôzny priebeh duktility pre jednotlivé variácie spojenia. Následne boli zistené hodnoty únosnosti porovnané so štandardným výpočtom únosnosti takéhoto spojenia podľa literatúry a normy.*

Kľúčové slová

Celozávitové skrutky, Duktilita, Experimentálne testovanie, Lepené lamelové drevo, Rámový roh

1. Úvod

Drevo sa ako stavebný materiál v stavebníctve stále viac dostáva do popredia pri stavbe rôznych konštrukcií. Tento trend je z časti zapríčinený aj vďaka upriameniu pozornosti na obnoviteľné zdroje. Dopyt po dreve ako stavebnom materiály ponúka možnosť zlepšenia jeho mechanických a fyzikálnych vlastností a preto sú vyvíjané nové kompozitné materiály na báze dreva alebo sa skúmajú zlepšenia spojenia jednotlivých segmentov pre zabezpečenie vyššej únosnosti a tuhosti konštrukcie. Príspevok sa preto zameriava na inováciu dreveného rámového spojenia vytvoreného pomocou mechanických spájacích prostriedkov kolíkového typu. Konkrétne boli namiesto štandardnej kombinácie svorníkov a kolíkov použité celozávitové skrutky. Jednotlivú korešpondenciu výsledkov experimentálneho testovania je potom možné porovnať s výsledkami dosiahnutých pomocou normy [1] a odbornej literatúry [2]. Príspevok je pokračovaním výskumnej činnosti autora na predchádzajúcu uverejnenú prácu [3].

2. Záver

V príspevku boli predstavené výsledky zistené experimentálnym testovaním rámového spojenia. Testovanie pozostávalo z troch variácií konštrukčného systému. Spevnenie jednotlivých segmentov ukazuje očakávane významný vplyv na duktilitu spojenia, pričom únosnosť zostáva pomerne nezmenená.

Poďakovanie

Tento príspevok bol financovaný zo Študentskej grantovej súťaže VŠB – TUO. Registračné číslo projektu je SP2021/120. Experimentálne merania boli realizované vďaka Katedre konstrukcí 221 VŠB – TUO, menovito: Ing. Pavel Dobeš a vďaka Experimentálnímu a diagnostickému stavebnímu centru 207, menovito: Ing. Petr Mynarčík, Ph.D.

Referencie

[1] ČSN EN 1995-1-1. Eurokód 5: Navrhovaní dřevěných konstrukcí – Část 1-1 Obecná pravidla – Společná pravidla a pravidla pro pozemní stavby. Praha: ÚNMZ, prosinec 2006

[2] Dřevěné konstrukce podle Eurokódu 5, STEP 1: navrhování a konstrukční materiály. Přeložil Bohumil Koželouh. Zlín: KODR, c1998. ISBN 80-238-2620-4

[3] Johanides, M.; Kubíncová, L.; Mikolášek, D.; Lokaj, A.; Sucharda, O.; Mynarčík, P. Analysis of Rotational Stiffness of the Timber Frame Connection. *Sustainability* **2021**, *13*, 156. https://doi.org/10.3390/su13010156

FATIGUE BEHAVIOUR OF A SHORT CRACK IN A THIN PROTECTIVE LAYER LASER-CLADDED ON A STEEL SUBSTRATE

Tereza JUHÁSZOVÁ[1,2], *Lucie MALÍKOVÁ*[1,2], *Pavel DOUBEK*[1,3], *Petr MIARKA*[1,2,4], *Stanislav SEITL*[1,2]

[1]Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Veveří 331/95, Brno, Czech Republic

[2]Institute of Physics of Materials, Czech Academy of Sciences, v. v. i., Žižkova 513/22, Brno, Czech Republic

[3]OMNI-X CZ s.r.o., Šámalova 60a, Brno, Czech Republic

[4]Department of Structural Engineering and Building Materials, Faculty of Engineering and Architecture, Ghent University, Technologiepark Zwijnaarde 60, Gent, Belgium

tereza.juhaszova@vutbr.cz, malikova.l@fce.vutbr.cz, pavel.doubek@omni-x.cz, petr.miarka@vut.cz, seitl@ipm.cz

Combinations of two different materials in structural elements are more and more common in engineering applications in order to improve properties of components. One of the popular technologies is the laser cladding [1].

In this contribution, the fatigue crack propagation through the laser-cladded layer is assessed. Surface layers made of materials with various Young's modulus are considered and their effect on the fracture parameters is investigated. The numerical simulations presented in this work are a part of a more extensive analysis that should finally bring recommendations on the proper layer thickness choice and material choice under a particular kind of loading with regard to fatigue lifetime of real components. A specimen prepared for oncoming experimental campaign is shown in Fig. 1.

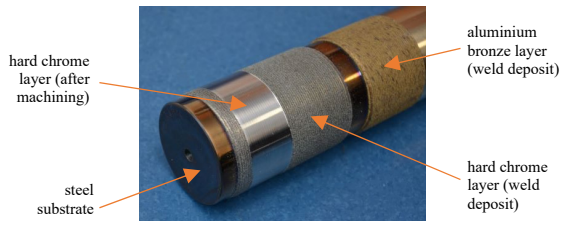


Fig. 1: Real cylindrical specimen consisting of a steel substrate coated with a hard chrome layer and/or an aluminium bronze layer.

In order to investigate the crack behaviour in various kinds of protective layers, numerical calculations via FEM were performed. A steel bar with thin surface layers on its both sides were modelled. One crack on each side of the specimen in its middle was considered and the specimen was subjected to 1) pure tension; 2) pure bending (couple-moment). The stress intensity factor range was investigated for different elastic modulus of the cladded layer (simulating various materials of the layer, such as tool steel, stainless steel types, cobalt, copper, nickel, or aluminium alloys). Results obtained can be seen in Fig. 2.

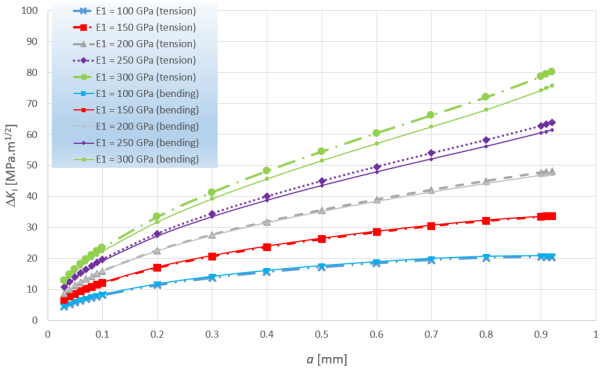


Fig. 2: Stress intensity factor range for various Young's modulus of the laser-cladded layer for two types of loading.

Based on the plots presented in Fig. 2 and knowledge of basic fatigue properties of the materials (threshold value of the stress intensity factor ΔK_{th} and critical value ΔK_{IC}), conclusions can be stated with regard to assessment of stable/unstable fatigue crack propagation. Thus, choice of the proper choice of the material of the layer and/or its thickness under a particular loading can be recommended.

Acknowledgements

Financial support from the Faculty of Civil Engineering, Brno University of Technology is gratefully acknowledged (project No. FAST-S-21-7338).

References

[1] ZHU, L., XUE, P., LAN, Q., MENG, G., REN, Y., YANG, Z., Xu, P. and Z. LIU. Recent research and development status of laser cladding: A review. *Optics & Laser Technology*. 2021, vol. 138, 10paper nr. 6915. ISSN 0030-3992. DOI: 10.1016/j.optlastec.2021.106915.

ANALYSIS OF STRESS FIELD SHAPE AROUND CRACK TIP OF STAINLESS-STEEL IPE PROFILE

Tereza JUHÁSZOVÁ[1,2], *Petr MIARKA*[1,2,3], *Stanislav SEITL*[1,2]

[1] Institute of Physics of Materials, Czech Academy of Sciences, Žižkova 22, Brno 616 62, Czech Republic
[2] Faculty of Civil Engineering, Brno University of Technology, Veveří 331/95, Brno 602 00, Czech Republic
[3] Department of Structural Engineering and Building Materials, Faculty of Engineering and Architecture, Ghent University, Technologiepark Zwijnaarde 60, Gent, Belgium

juhaszova@ipm.cz, miarka@ipm.cz, seitl@ipm.cz

Use of stainless steel as a structural material is highly common nowadays. Thanks to its *great* resistance to *aggressive environment, high yielding* strength, flexibility and good weldability, it can be widely found *used* in large structures like bridges and water supply systems. In comparison to other structural materials various geometry and angle connections of elements are more accessible. On the other hand, it has also an attractive appearance, therefore it can also be used in exterior structural work like roofing and barriers, handrails and for the entrance structures.

Many of loadbearing stainless steel elements are subjected to cyclic load which can lead to fatigue damage. Linear elastic fracture mechanic analysis is performed on IPE profile, which is beam component typical for steel structures. 3D parametrical study is processed using FEM software Ansys Mechanical, specimen model with crack in software is displayed in Figure 1.

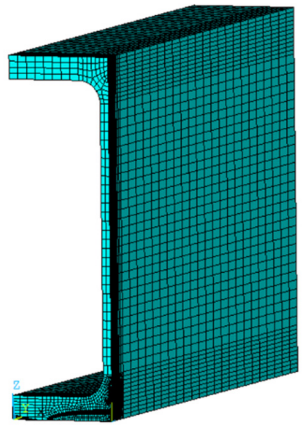


Fig. 1: 3D numerical model of IPE profile.

Stress field around crack tip is modelled in variations of shapes and the stress intensity factor (SIF) is the main characteristic used to describe the stress field distribution around crack tip. Geometries of crack front shapes are presented in Figure 2.

The only half of the IPE profile was modelled to symmetry conditions. The specimen was loaded to represent three- point bending and the results obtained on 3D model are calibrated and compared with 2D solution.

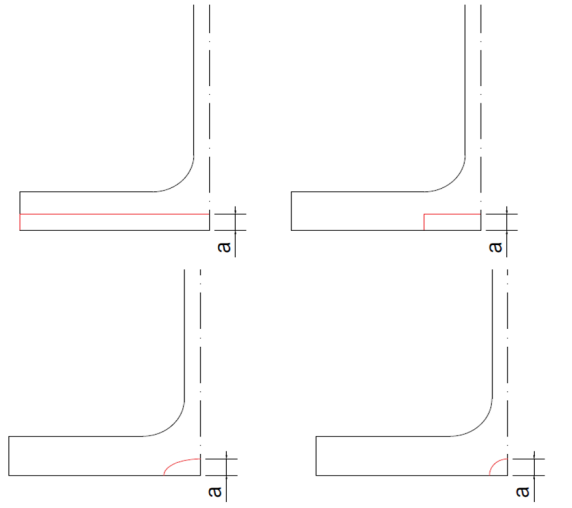


Fig. 2: Sketch of crack front shapes configurations

Results of this numerical parametric study are further investigated in order to obtain geometry of the crack front which would plot values of SIF approximated to 2D solution and 3D specimen with rectangular cross- section.

Acknowledgements

The financial support of the grants No. FAST-J-21-7340 and from GACR project no. 20-00761S is greatly appreciated.

Proof of concept off two-layered dome-like structure for lunar bases

Petr KONEČNÝ[1], *Jacek KATZER*[2]

[1]Department of Structural Mechanics, Faculty of Civil Engineering, VSB – Technical University of Ostrava, L. Podéště 1875, Ostrava, Czech Republic
[2]Center of Civil Engineering, Faculty of Geoengineering, University of Warmia and Mazury in Olsztyn, Jana Heweliusza 4, Olsztyn, Poland

petr.konecny@vsb.cz, jacek.katzer@uwm.edu.pl

Abstract. *The paper is dealing with preliminary analysis of the concept off two-layered dome-like structure for lunar bases with respect to its the stability. There are constraints that makes it impossible to bring the traditional source materials and technology for the reinforced concrete construction of Lunar and Martian bases. Therefore the article is dealing with one of the possible strategies to overcome this limitation. The proof of concept of the structural shell of the dome based on the coarse fraction regolith-based concrete-like material and protection overlay material from the fine fraction of regolith is preliminary evaluated with respect to stability of proposed solution. The initial numerical analysis of the stability of the dome with respect to dead-load and the inside-air caused uplift via direct stiffness method was conducted.*

Keywords

ISRU, Moon, Dome, Sandwich, Habitat, Model, Stability.

1. Introduction

The human endeavor to build a bases and to live on Luna and Mars tempted mankind for almost two centuries. There is a question, though, what construction material may be used. The concrete that is among the most common one on Earth seems to be good choice. However, it is obvious that transportation of all the construction material from Earth to the construction site is not possible. Therefore, the in-situ resource utilization (ISRU) and automation shall replace the traditional earth materials such as aggregates and sand as well as human work. Moreover, the attention might be driven to the fully automated construction process including 3D printed reinforcement.

The dome-like shell seems to be most effective structural solution having in mind the pressure from the air inside the dome. Moreover, the meteorites impacts and the significant temperature changes effect are potential risk to the structural integrity of the structure air-tightness of it. Therefore, the sandwich-like structure is proposed: the structural shell controls the stability and the soil overlay protects from the climate effects. Moreover, the balloon like support and lost form work is considered as well in order to maintain the air-tightness. The concrete-like material might be prepared from the coarse fraction of the regolith and the overlay from the fine fraction. The coarse fraction would be separated by magnetic separation.

The paper is dealing with preliminary analysis of the stability of the concept off two-layered dome-like structure for lunar bases. The material from the shell of the dome is intended to be from the structural coarse fraction regolith-based concrete-like material and protection overlay from the fine fraction of regolith. The preliminary numerical analysis of the stability of the dome with respect to dead-load and the inside-air caused uplift via direct stiffness method is conducted and discussed in the full paper.

Acknowledgment

Financial support from VSB-Technical University of Ostrava by means of the Czech Ministry of Education, Youth and Sports, project *Science without borders 2.0* Nr. CZ.02.2.69/0.0/0.0/18_053/0016985 and co-funding by National Science Centre (Poland) grant number DEC-2020/38/E/ST8/00527 *Regolith harvesting on Moon surface: Excavation and beneficiation in low gravity environments* is gratefully acknowledged.

FREQUENCY DETERMINATIONS OF FLUID FILLING AND MONITORING OF SLOSHING PROBLEM IN RECTANGULAR CONTAINER

Kamila KOTRASOVA[1], *Eva KORMANIKOVA*[2]

[1]Institute of Structural Engineering, Technical University of Kosice, Vysokoskolska 4, 042 00 Kosice, Slovakia
[2]Institute of Structural Engineering, Technical University of Kosice, Vysokoskolska 4, 042 00 Kosice, Slovakia

kamila.kotrasova@tuke.sk, eva.kormanikova@tuke.sk

Abstract. *The sloshing problem is relevant to the analysis of liquid transportation, the study of the effect of vibrations on liquid storage tanks or liquid damper devices, and many other engineering applications. Several methods have been developed to study and predict free surface behaviour, such as experimental studies, analytical solutions, and numerical methods. This work described analytical, numerical, and experimental possibilities of monitoring of frequency determination of fluid filling and sloshing problems in liquid storage tanks.*

Keywords

Fluid, Natural Frequency

1. Introduction

Ground-supported tanks are used to store a different kind of liquids. The motion of the container, full or partially filled with liquid, causes the hydrodynamic pressure and fluid flow up to sloshing of free surface and forms the basis for many complex problems. The free liquid surface may experience different types of motion including simple planar, non-planar, rotational, irregular beating, symmetric, asymmetric, quasi-periodic and chaotic, it is depended on the type of excitation and container shape. The amplitude of slosh depends on the frequency and amplitude of the tank motion, liquid-fill depth, liquid properties, and tank geometry. The fluid resonance in the case of horizontal excitation occurs when the external forcing frequency is close to the natural frequency of the liquid. The liquid sloshing is a practical problem with regard to the reliability and safety structures, because an eventual damage of containers used for storage of hazardous liquids, as petroleum, chemical and radioactive waste are catastrophic, consequences are financial, and environmental loses.

2. Natural frequency analysis

2.1. Analytical solution of natural frequencies

The determination of the n-th natural angular frequency ω_n can be obtained from the analytical equations according to Housner, which applies to a rectangular tank

$$\omega_n^2 = n\frac{1{,}58g}{L}\,tanh\left(n1{,}58\frac{H}{L}\right) \quad (1)$$

The n-th natural angular frequency ω_n for a cylindrical tank can be determined

$$\omega_n^2 = n\frac{1{,}84g}{R}\,tanh\left(n1{,}84\frac{H}{R}\right) \quad (2)$$

where R is the radius of the cylindrical tank, L is half the length of the rectangular tank, and H is height of fluid filling.

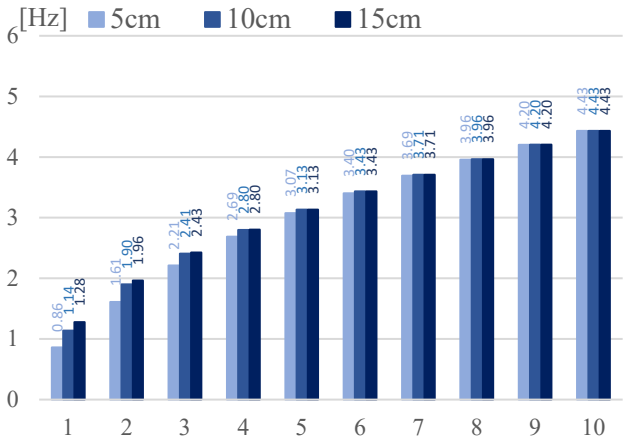


Fig. 1: The comparison of the first ten natural frequencies of fluid filling for 5 cm, 10 cm, and 15 cm filling

This work was supported by the Project VEGA 1/0374/19.

The prediction of fatigue life of notches

Kamila KOZÁKOVÁ[1], *Jan KLUSÁK*[1]

[1]Institute of Physics of Materials, Czech Academy of Sciences,
Žižkova 512/22, 616 00 Brno, Czech Republic

kozakova@ipm.cz, klusak@ipm.cz

Abstract. *The contribution presents a method for lifetime predictions of components with notches. The method was developed using experimental data and numerical analysis of notched specimens. The method employs approaches of generalized fracture mechanics. Emphasis is placed on the area of gigacycle fatigue.*

Keywords

Wöhler curve, fatigue life, notches, generalized fracture mechanics, stress distribution.

Method

Computational methods bring the possibility of precise analysis of stress concentrations in notches. Fatigue life curves are usually determined for smooth specimens and rarely for notched samples. Two sets of specimens (notched and smooth) from structural steel were tested in the area of high and very high cycle fatigue [1]. In order to estimate fatigue life of specimens with different notch radii an approaches of fracture mechanics were employed. The suggested method forms relation between the life curves of smooth and notched specimens and uses distribution of the axial stress (fig. 1).

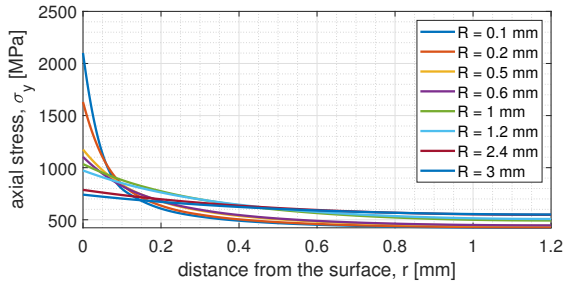


Fig. 1: Axial stress distribution

The line method of the theory of critical distances [2], was used to find the relationship between the S-N curves of smooth and notched samples (R = 0.1 mm).

$$\sigma_{y,\mathrm{LM}}(l) = \frac{\sigma_{\mathrm{a,nom}}^{\mathrm{smooth}}(N_\mathrm{f})}{\sigma_{\mathrm{a,nom}}^{\mathrm{notched}}(N_\mathrm{f})} \cdot \sigma_{y,\mathrm{LM,\ nom}} \qquad (1)$$

The output of the equation (1) is the critical length parameter l. We assume that this length parameter is a material property and can be used to predict the lifetimes of specimens with different notch radii.

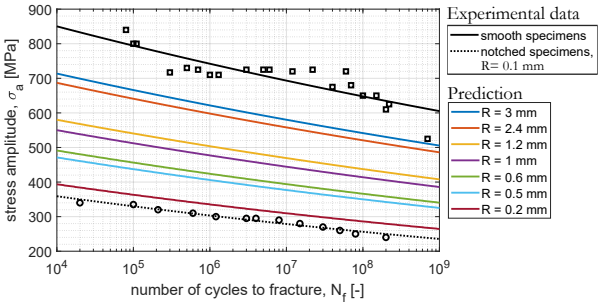


Fig. 2: Fatigue data and predictions of lifetime curves

Conclusion

Using the precise analysis of stress distribution of the notched specimens and two lifetime curves (notched and smooth specimens), it is possible to predict lifetime of specimens with different notch radii. The average critical distance for the steel is 0.08 mm and slightly decreases with the increasing number of cycles to fracture.

Acknowledgment

The research was supported by the Czech Science Foundation, project No. 21-14886S.

References

[1] Qian, G., Hong, Y., Zhou, C.: Investigation of high cycle and very-high-cycle fatigue behaviors for a structural steel with smooth and notched specimens. Eng. Fail. Analysis 17(7), 1517–1525 (2010)

[2] Taylor, D.: The theory of critical distances: A link to micromechanisms. Theoretical and Applied Fracture Mechanics 90, 228–233 (2017)

PROBABILISTIC NONLINEAR ANALYSIS OF STEEL FRAME SAFETY UNDER EXTREME WIND IMPACT

Juraj KRÁLIK[1], *Juraj KRÁLIK, jr.*[2]

[1]Department of Structural Mechanics, Faculty of Civil Engineering, Slovak University of Technology, Radlinského 11, Bratislava, Slovakia
[2]Institute of Structures in Architecture and Engineering Buildings, Faculty of Architecture and Design, Slovak University of Technology, Námestie slobody 19, Bratislava, Slovakia

juraj.kralik@stuba.sk, ing.kralik@hotmail.com

Abstract. *This paper describes the methodology and the results of the safety analysis of the Nuclear Power Plant structures under impact of the extreme climatic loads. The requirements of the international agency IAEA and NRC standards are based on the probability of mean return period equal to one per 10^4 years. This probabilistic assessment of NPP structures for Probabilistic Safety Analysis (PSA) level 2 of VVER 440/213 in the case of the extreme wind impact is considered. The experimental and numerical analysis of the wind load are presented.*

Keywords

Safety, reliability, fragility, nuclear power plant, probability, experiment, wind, ANSYS, FLUENT, FReET.

1. Introduction

This paper presents the reliability analysis of the steel hall frame of the NPP critical transversal gallery resistance due to extreme wind loads. The extreme loads were defined for mean return period equal to one per 10^4 years in accordance with IAEA requirements for NPP structures. The new methodology of the safety and reliability analysis of the NPP structures are presented. The extreme wind impact to the NPP buildings was considered in wind tunnel of STU Bratislava. The models of the NPP buildings were considered without and with the surrounding buildings. The objects were assembled on a 3D printer in a scale of 1: 300. After the measurement in the wind tunnel, it was necessary to verify the measured values on a 3D model of wind flow on individual critical surfaces. The errors can also occur in defining the boundary conditions as well as in the computational processes during the simulation and processing of the results in the postprocessor. The resulting pressures from extreme winds were therefore considered as the best estimate corresponding to the average value of the reduced peak pressures (according to the Eurocode).

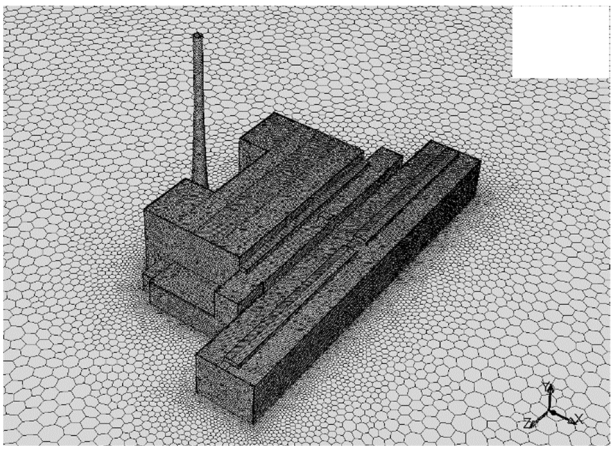


Fig.1: 3D FE fluid model in software ANSYS-FLUENT.

The new presented design methodology is based on the proposition that the failure of the structures is determined by the failure of the principal critical structural systems. The principal problem of the wind resistance of the NPP structures was that the real pressures of the wind impact is diametral indifferent in comparison of the Eurocodes. The geometric and material nonlinearity were considered. The deterministic and probabilistic analysis of the structure failure were investigated. The results of this analysis clearly confirm that considered NPP steel structures have significantly greater resistance to extreme wind effects at the origin considering the development of plastic deformations.

References

[1] KRÁLIK, J. *Safety and Reliability of Nuclear Power Buildings in Slovakia. Earthquake-Impact-Explosion*. Edition STU Bratislava, 305 p. 2009.

Probabilistic Prediction of Fatigue Damage of HSS Structures

Martin KREJSA[1], *Jiri BROZOVSKY*[1], *Petr LEHNER*[1]

[1]Department of Structural Mechanics, Faculty of Civil Engineering, VSB-Technical University of Ostrava, Ludvika Podeste 1875/17, 708 00 Ostrava–Poruba, Czech Republic

martin.krejsa@vsb.cz, jiri.brozovsky@vsb.cz, petr.lehner@vsb.cz

Abstract. *In the construction of steel structures and especially bridges, the use of high-strength steel (HSS) has been increasing in recent decades. The use of steel with a higher yield strength allows the design of slimmer as well as more aesthetic constructions. Design often leads to cost and material savings. In cases where the ultimate limit state of the structure is not a decisive factor of the design (but on the contrary the serviceability limit state, dynamic behavior of the structure, stability or fatigue of the material), the advantage of HSS becomes questionable. In such cases, the resulting resistance of the HSS structure is affected mainly by fatigue resistance. The article is focused on the initial study of fatigue damage prediction of HSS structures using linear fracture mechanics and probabilistic approach.*

Keywords

Fatigue, steel, HSS, failure.

1. Introduction

In the construction of steel structures and especially bridges, the use of high-strength steel (HSS - steel with a yield strength of 460 MPa or higher) has been increasing in recent decades. The use of steel with a higher yield strength allows the design of slimmer as well as more aesthetic constructions. Design often leads to cost and material savings. In cases where the ultimate limit state of the structure is not a decisive factor of the design (but on the contrary the serviceability limit state, dynamic behavior of the structure, stability or fatigue of the material), the advantage of HSS becomes questionable. In such cases, the resulting resistance of the HSS structure is affected mainly by fatigue resistance, because reducing the cross-section of the structure leads to a higher concentration of stress in the details prone to fatigue damage. For this reason, it is a very current topic for research, which is addressed by many authors, and this is the reason that issues associated with the use of HSS have to be given due attention.

Fatigue degradation of high strength steel is likely to be different than that of low/medium steel and not sufficiently explained yet and needs further investigation. Propagation of the fatigue cracks and possible prediction of such propagation in time since the start of variable loading effects is the case when probabilistic methods must be used because too many uncertainties influence the determination of the input values. Stochastic approach is a powerful computing tool, which brings with it some difficulties associated mainly with insufficiently relevant input data and also often with computational complexity. Probabilistic analysis of structural durability usually leads to estimate the lifetime of the analyzed load-bearing elements. For this reason, new probabilistic approaches are still evolving. One of them is also the DOProC method, which is an important method for this type of calculation. This approach is characterized by the high precision of computing of the failure probability p_f, as well as considerable efficiency of the calculation in a number of probability tasks. Presented analysis is focused on the initial study of fatigue damage prediction of HSS structures using DOProC method and linear fracture mechanics.

Acknowledgment

This contribution has been developed as a part of the research project GACR 21-14886S "Influence of Material Properties of High Strength Steels on Durability of Engineering Structures and Bridges" supported by the Czech Grant Agency and also has been completed thanks to the financial support provided to VSB-Technical University of Ostrava by the Czech Ministry of Education, Youth and Sports from the budget for conceptual development of science, research and innovations for the 2021 year.

POSOUZENÍ KOROZNĚ POŠKOZENÝCH SPOJŮ OCELOVÝCH KONSTRUKCÍ NAVRŽENÝCH Z PATINUJÍCÍCH OCELÍ

Vít KŘIVÝ[1], *Miroslav VACEK*[1], *Zdeněk VAŠEK*[2]

[1]Fakulta stavební, VŠB – Technická univerzita Ostrava, Ludvíka Podéšte 1875/17, Ostrava - Poruba
[2] Liberty Ostrava a.s., Vratimovská 689/117, 719 00 Ostrava - Kunčice

vit.krivy@vsb.cz, miroslav.vacek@vsb.cz, zdenek.vasek@libertysteelgroup.com

Abstract. *Corrosion of steel lattice tower joints is a structural and economical issue. The aim of this article is the corrosion degradation of bolted joints of steel lattice towers made from weathering steel. Attention is paid especially to the effect of crevice corrosion on the load-bearing capacity of the joint. The measured data from the implemented loading tests are compared with the results of analytical and numerical models.*

Klíčová slova

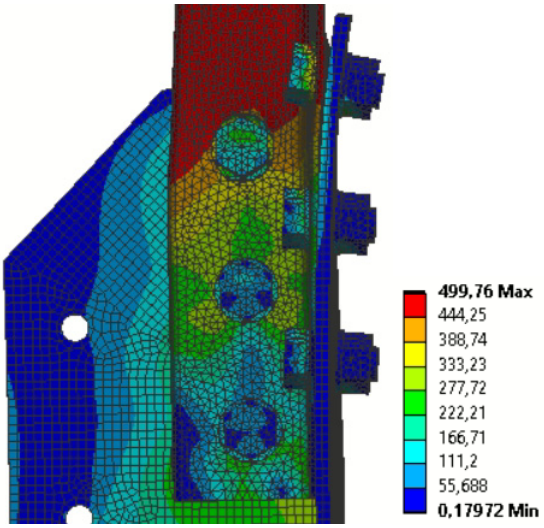
Spárová koroze, styčník, stožár elektrického vedení, havárie, Atmofix, příložky, patinující ocel

1. Úvod

Patinující oceli jsou v České republice využívány pro realizaci ocelových konstrukcí již od poloviny 70. let minulého století. Typickými objekty jsou především mosty a stožáry. Ocelové příhradové stožáry se obvykle používají jako nosné konstrukce v energetických přenosových soustavách. Tyto konstrukce představují nejrozsáhlejší realizaci patinujících ocelí v České republice. V letech 1974 až 1992 bylo postaveno přibližně 4000 stožárů vedení a 130 rozvoden napětí 110 kV, 220 kV a 400 kV. Příhradové stožáry elektrického vedení jsou vyráběny v různých tvarových variantách, vždy ve tvaru čtyřbokých příhradových konstrukcí, vyztužených v jednotlivých stěnách diagonálami a příčkami z jednoduchých úhelníků. Nárožníky z úhelníků jsou po výšce stykovány pomocí šroubových spojů s příložkami, úhelníky ztužení jsou k nárožníkům připojeny pomocí jednoho šroubu.

Korozní chování patinujících ocelí na elektrovodních a vysílacích stožárech je systematicky hodnoceno od roku 1975. Obecně lze konstatovat, že na nosných prvcích stožárů dochází k příznivému vývoji ochranných korozních produktů. Pro použití patinujících ocelí však byly uplatněny stejné detaily jako pro stožáry metalizované zinkem nebo chráněné nátěrem, pro provozní kontrolu a údržbu stožárů nebyly specifikovány žádné zvláštní požadavky. V nevhodně konstruovaných detailech proto dochází k nepříznivému vývoji korozních produktů. U šroubového spoje nárožníků dochází ve spáře mezi příložkami a spojovanými úhelníky ke vzniku spárové koroze. Ve spáře se prodlužuje doba ovlhčení povrchu, mohou se hromadit nečistoty, vznikají aerační a koncentrační rozdíly, což vše podporuje rozvoj korozního procesu. Ve spáře tak nenastávají podmínky potřebné pro vznik ochranné patiny. Nahromaděné korozní produkty deformují příložky ve spoji. Obava z možných účinků nepříznivého korozního vývoje ve spojích nárožníků vedla energetické společnosti v České republice k opuštění konceptu uplatnění patinujících ocelí při výstavbě nových linek elektrického vedení. Aby bylo možné stávající stožáry zodpovědně provozovat, je důležité mít k dispozici ověřené technické údaje o vlivu spárové koroze na únosnost styčníků.

V článku jsou proto uvedeny výsledky zatěžovacích zkoušek styčníků nárožníků stožárů elektrického vedení 2x110 kV, které byly odebrány z havarovaných stožárů na vedení *Neznášov – Týniště nad Orlicí*. Hlavním cílem experimentálního programu a následných výpočtových analýz, viz obr. 1, bylo ověření vlivu spáry mezi jednotlivými prvky spoje na mechanickou odolnost konstrukce stožáru navrženého z patinující oceli.



Obr. 1 Srovnávací napětí [MPa] na styčníku typu A

CORRELATION BETWEEN SURFACE AND BULK RESISTIVITY ON PRISM SPECIMENS FROM LUNAR AGGREGATE SIMULANT

Petr LEHNER[1], *Petr KONEČNÝ*[1], *Jacek KATZER*[2]

[1]Department of Structural Mechanics, Faculty of Civil Engineering, VSB – Technical University of Ostrava, Ostrava, Czech Republic
[2]Center of Civil Engineering, Faculty of Geoengineering, University of Warmia and Mazury in Olsztyn Olsztyn, Poland

petr.lehner@vsb.cz, petr.konecny@vsb.cz, jacek.katzer@uwm.edu.pl

Abstract. *The field of space engineering is attracting increasing attention as the time approaches when it will be necessary to colonize our nearest space neighbour quickly and efficiently. The huge challenge is to change the established procedures suitable for our home planet and also to make the most of resources without the need for disproportionately complicated space transportation. The aim of this work is to expand our knowledge of the lunar aggregate simulator (LAS) based on ilmenite rock, which is available in Central Europe, and which has good and suitable properties. Prismatic samples from mortar based on standard sand and ilmenite were prepared. The measurements were collected using several different destructive (DT) and non-destructive (NDT) methods, and the results were compared, and mutual correlation was evaluated. The presented procedure for experimental testing combining DT and NDT methods in one sample represents a potential streamlining of processes.*

Keywords

Compressive strength, lunar aggregate simulant, concrete, electrical resistivity, correlations.

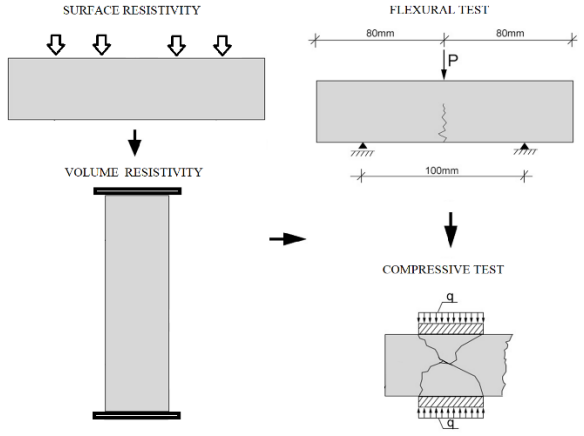


Fig. 1: The process of using the samples for surface resistivity, volume resistivity, flexural test, and compressive strength.

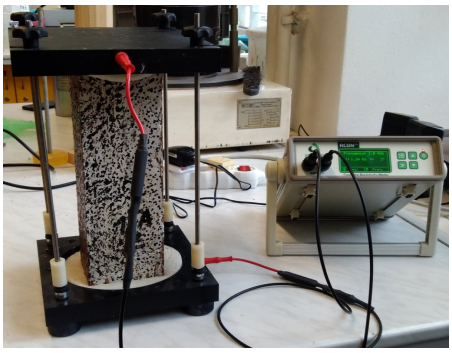


Fig. 2: Example of measuring volume resistivity of a prismatic sample.

This work presented the application of the known measurement of electrical resistance to a new material serving as lunar aggregate stimulants. The electrical resistance from two different tests was compared with elementary methods for determining strength. The most important limitation of the results is the fact that we do not have enough information about the behaviour of rocks on the moon. Current findings could help solve this problem because they have shown the limitations and similarities with known composite materials. These results provide a significant step towards the impending colonization of space, which will need support in the form of building development.

Acknowledgements

Financial support from VSB-Technical University of Ostrava by means of the Czech Ministry of Education, Youth and Sports through the Institutional support for conceptual development of science, research and innovations for the year 2021 is gratefully acknowledged. This research was partially funded by National Science Centre (Poland), grant number DEC-2020/38/E/ST8/00527 (Regolith harvesting on Moon surface: Excavation and beneficiation in low gravity environments).

3PBT AND WST OF MULTIPLE SPECIMEN SIZES: MECHANICAL FRACTURE PARAMETERS OF C30/37 CONCRETE BY INVERSE ANALYSIS

Martin LIPOWCZAN, Martina ŠOMODÍKOVÁ, David LEHKÝ

Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Veveri 95, Brno 602 00, Czech Republic

lipowczan.m@fce.vutbr.cz, somodikova.m@fce.vutbr.cz, lehky.d@fce.vutbr.cz

Abstract

The present contribution summarizes the identification part of the project concerned with the development of a comprehensive multilevel approach for the experimental–computational determination of the mechanical fracture parameters of C30/37 concrete. Three-point bending tests (3PBT) and wedge-splitting tests (WST) were performed simultaneously using three different geometrically similar specimen sizes in the ratio 1:3 and two well-separated depths of initial notches. Examples of both test configurations are depicted in Fig. 1.

Fig. 1: Test configurations of a three-point bending test (left) and wedge-splitting test (right).

Experiments were followed by identification of the mechanical fracture parameters using inverse analysis based on an artificial neural network [1]. The inverse procedure transforms fracture test response data into the desired mechanical fracture parameters. This approach is based on matching laboratory measurements with the results gained by reproducing the same test numerically.

Three material parameters were subject of identification – modulus of elasticity E, tensile strength f_t, and fracture energy G_f. The mechanical fracture parameters were successively identified for all 36 specimens tested in 3PBT and WST configurations. Resulting mean values (coefficients of variation) of C30/37 concrete, obtained from all tested 3PBT and WST specimens, were: $E = 31.42$ GPa (12.8 %), $f_t = 3.12$ MPa (18.7 %), $G_f = 192.85$ J/m^2 (17.7 %).

When comparing the values of the parameters obtained for different specimen sizes, a certain degree of dependence on the size of the initial uncracked ligaments was evident. Fracture energy value is influenced by the size of the fracture process zone, which in turn is influenced by the free boundary of the test specimen. Figure 2 depicts values of fracture energy obtained from 3PBT plotted against the depth of the initial uncracked ligament. A size dependence was also confirmed for the tensile strength. On the other hand, the results confirmed the size independence of the modulus of elasticity.

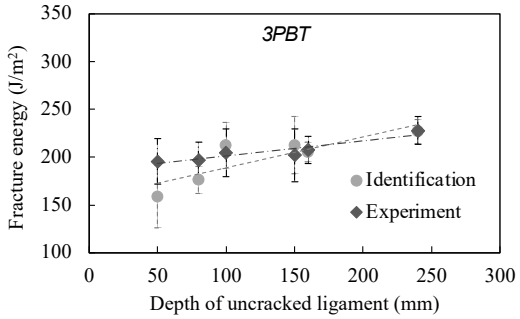


Fig. 2: Dependence of fracture energy on the size of the uncracked ligament: identification vs. experiment for 3PBT specimens.

Acknowledgements

The authors would like to express their thanks for the support provided from the Czech Science Foundation project MUFRAS No. 19-09491S and the specific university research project No. FAST-J-21-7552 granted by Brno University of Technology.

References

[1] NOVÁK, D. and D. LEHKÝ. ANN Inverse Analysis Based on Stochastic Small-Sample Training Set Simulation. *Engineering Application of Artificial Intelligence*. 2006, vol. 19, pp. 731–740. DOI: 10.1016/j.engappai.2006.05.003.

DYNAMIC RESPONSE OF A CONCRETE SLAB

Jozef MELCER[1]

[1]Department of Structural Mechanics and Applied Mathematics, Faculty of Civil Engineering, University of Žilina, Univerzitná 8215/1, 010 26 Žilina, Slovakia

jozef.melcer@uniza.sk

Keywords

Numerical analysis, moving load, concrete slab, Winkler's model, Pasternak's model.

1. Vehicle Computational Model

For the purpose of this paper the so called quarter model of vehicle was adopted. Equations of motion for calculation of unknown functions, describing the vertical displacements of lumped masses m_i, can be written as

$$\ddot{r}_1(t) = \{-k_1 d_1(t) - b_1 \dot{d}_1(t)\}/m_1,$$
$$\ddot{r}_2(t) = \{+k_1 d_1(t) - k_2 d_2(t) + b_1 \dot{d}_1(t) - b_2 \dot{d}_2(t)\}/m_2. \quad (1)$$

The term for calculation of contact force is

$$F(t) = -G_2 + k_2 d_2(t) + b_2 \dot{d}_2(t). \quad (2)$$

The values $d_i(t)$, (i = 1, 2) represent the deformation of connected members of the model in time t. The derivations with respect to time are denoted by the dot above the symbol of dependently variable.

2. Computational Model of a Concrete Slab

In the spirit of classical mechanics the slab computational model is created in the sense of Kirchhoff theory of thin slabs on elastic foundation of Winkler and Pasternak type.

The equation of motion describing the slab vibration on Winkler foundation has the form

$$D\left(\frac{\partial^4 w}{\partial x^4} + 2\frac{\partial^4 w}{\partial x^2 \partial y^2} + \frac{\partial^4 w}{\partial y^4}\right) + Cw + \mu\frac{\partial^2 w}{\partial t^2} + 2\mu\omega_b\frac{\partial w}{\partial t} = p(x,y,t). \quad (3)$$

In case of Pasternak foundation the equation of motion is

$$D\left(\frac{\partial^4 w}{\partial x^4} + 2\frac{\partial^4 w}{\partial x^2 \partial y^2} + \frac{\partial^4 w}{\partial y^4}\right) + C_1 w - C_{2X}\frac{\partial^2 w}{\partial x^2} - C_{2Y}\frac{\partial^2 w}{\partial y^2} + \mu\frac{\partial^2 w}{\partial t^2} + 2\mu\omega_b\frac{\partial w}{\partial t} = p(x,y,t). \quad (4)$$

The assumption about the shape of the bending area of the slab $w_a(x,y)$ due to the load is adopted and the wanted function $w(x,y,t)$ describing the shape of the bending area of the slab in time t is expressed as

$$w(x,y,t) = q(t)w_a(x,y) = q(t)\sin\frac{\pi x}{l_x}\sin\frac{\pi y}{l_y}. \quad (5)$$

I case of discrete moving load the contact force $F_j(t)$ must be transformed on continuous load $p(x,y,t)$ by [1] using the Dirac´s δ function

$$p(x,y,t) = \sum_j F_j(t)\delta(x - x_j)\delta(y - y_j). \quad (6)$$

3. Pavement Composition

The subject of the analysis is the concrete pavement on the layered subsoil. For the subsoil the constants C, C_1, C_{2X}, C_{2Y}, which characterize its properties in the spirit of Winkler's and Pasternak's model were calculated. We assume an isotropic subsoil $C_{2X} = C_{2Y} = C_2$.

4. Numerical Solution

Numerical solution of equations of motion was performed in MATLAB and the ode45 procedure was used [8]. In the first step, the Winkler's subsoil model is considered. In the second step the Pasternak's model is considered. For the Pasternak's model a correction of the results is needed due to the fact that the slab transmits to the subgrade only the value $k \cdot F$ of the contact force, where

$$k = 1/(1 + \frac{2K_1(\rho_0)}{\rho_0 K_0(\rho_0)}). \quad (7)$$

After these corrections, the model provides acceptable results. In the case of contact forces between the vehicle wheel and the slab, both models give practically identical results. A more precise model can be created only in the sense of FEM

Acknowledgements

This work was supported by Grant National Agency VEGA of the Slovak Republic. Project number G/0006/20.

INFLUENCE OF THICKNESS ON THE VALUES OF STRESS INTENSITY FACTOR

Petr MIARKA[1,2], *Stanislav SEITL*[1,2]

[1]Institute of Physics of materials, Czech Academy of Sciences, Žižkova 513/22 616 00, Brno, Czech Republic
[2]Institute of Structural Mechanics, Faculty of Civil Engineering Brno University of Technology; Veveří 331/95, 602 00 Brno, Czech Republic

miarka@ipm.cz, seitl@ipm.cz

Abstract. *This contributions deals with an assessment of the influence of the thickness on the values of stress intensity factor (SIF) measured on the compact-tension specimen. For this a numerical model was assembled and the SIF values were calculated for various specimen's thicknesses by direct method.*

Keywords

CT specimen, SIF, Thickness, Fracture.

1. Introduction

The compact tension (CT) specimen is commonly used in the experimental measurements of the fatigue crack growth rates in metallic materials by Paris' law [1]. Furthermore the standard for such measurements ASTM E647 [2] recommends the minimum thickness to achieve plane strain conditions and to have constant crack increment over the load cycles. Nonetheless the influence of the specimen's thickness has not been investigated in detail.

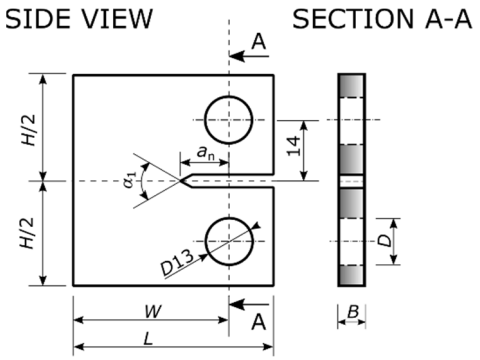


Fig. 1: Compact tension specimen

In case traditional size of $W = 50$ mm, the stress conditions for very thin specimen with a thickness $B < 2.5$ mm are believed to be as plane stress while for the specimens with thickness $B > 10$ mm it is acting as plane strain. However, the practical material sometimes require to prepare specimens with different thicknesses.

Based on this fact, a 3D numerical model in commercial finite element (FE) software ANSYS was created and the values of SIF were calculated by direct method. Furthermore, the specimen's thickness B varied from 1 ÷ 20 mm.

Acknowledgements

The financial support of the grants No. FAST-J-21-7340 and from GACR project no. 20-00761S is greatly appreciated.

References

[1] PARIS, P. and F. ERDOGAN. A Critical Analysis of Crack Propagation Laws, *Journal of Basic Engineering*, 1963, iss. 85, pp. 528-533. ISSN 0098-2202 DOI: 10.1115/1.3656900

[2] ASTM E647-15e1, Standard Test Method for Measurement of Fatigue Crack Growth Rates. In: ASTM International, West Conshohocken, PA; 2015

About Authors

Petr MIARKA was born in Český Těšín, Czech Republic. He received his Ph.D. from FCE BUT in 2021. His research interests include numerical simulation, fatigue and fracture testing of civil engineering materials.

Stanislav SEITL was born in Přerov, Czech Republic. He received his habilitation (assoc. prof.) from FCE BUT in 2015. His research interests include numerical simulation, fatigue and failure analysis and fracture-mechanical properties of civil engineering materials.

PRESSURE COEFFICIENT OF THE CYLINDER WITH CORRUGATED SURFACES

Vladimíra MICHALCOVÁ[1], *Ivan KOLOŠ*[1] *and Lenka LAUSOVÁ*[1]

[1]Department of Structural Mechanics, Faculty of Civil Engineering, VSB – Technical University of Ostrava, Ludvíka Podéště 1875/17, Ostrava-Poruba, Czech Republic

vladimira.michalcova@vsb.cz, ivan.kolos@vsb.cz, lenka.lausova@vsb.cz

1. Introduction

The work presents the role of numerical modelling of flow around a circular cylinder with a rough surface, which represents a smokestack. The 2D task simulates a highly turbulent flow corresponding to the real atmosphere. The calculations are carried out in the ANSYS Fluent software using the SST k-ω model, which allows a better mathematical description near the wall compared to other RANS models. The solution is examined on two basic types of roughness, which differ only in their density.

The study is focused on the calculation of the pressure load of the cylinder from the effect of the wind, which is represented here by the dimensionless pressure coefficient c_p. Since it is a very rough surface, attention is also paid to the influence of the local point location in surface roughness. Differences are observed not only in terms of local load, but also in its turbulent (fluctuation) characteristics. The drag coefficients c_d in both cases are compared to the standard calculation.

The flow around the cylinders is a complex phenomenon that is highly dependent on many parameters that can significantly affect flow field properties. Wind flow in the real atmosphere is a highly critical event in critical mode. Therefore, not only the Reynolds number Re (turbulent flow rate), but also the roughness (Fig. 1) and the irregularity of the flowing surfaces, play an important role in the loading of structures. The subject of this study is the effect of surface roughness on the flow field properties around the cylinder.

Fig. 1: Building structure with corrugated surface [https://tinyurl.com/chimney123].

The task is focused on modeling the flow around a cylinder with a corrugated surface. The paper describes a simulation of the wind flow around a real chimney with a diameter of 3.6 m. A diagram of two types of shaped surfaces is shown in Figures 2 and 3. In both cases the wave height is 18 mm. The main difference between the monitored profiles is their wave density. Flow parameters in the atmospheric boundary layer correspond to highly turbulent flow. In the study, the wind velocity is considered to be u_x = 10 m/s and the turbulence intensity i_u = 10 %. Reynolds number Re = $2.1 \cdot 10^6$.

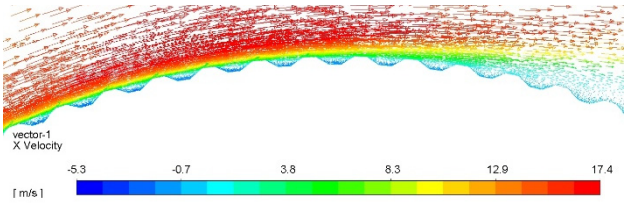


Fig. 2: Detail of geometry with dense waves on the perimeter (including velocity vectors around the cylinder).

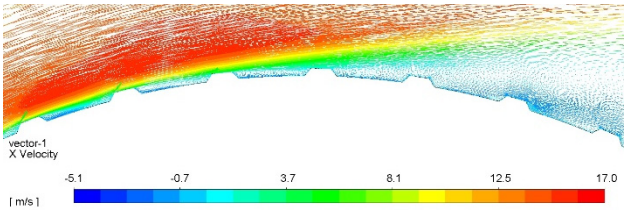


Fig. 3: Detail of sparse wave geometry on the perimeter (including velocity vectors around the cylinder).

Acknowledgements

In The paper has been supported by the project of "Conceptual development of science and research activities 2021" on the Faculty of Civil Engineering, VŠB – TU Ostrava and by the Ministry of Education, Youth and Sports from the Large Infrastructures for Research, Experimental Development and Innovations project „IT4Innovations National Supercomputing Center – LM2015070".

ANALYSIS OF THE INFLUENCE OF STRENGTH DISTRIBUTION ON WIRE CONCRETE

David Mikolášek[1], *Antonín Lokaj*[1], *Kristýna Vavrušová*[1], *David Bujdoš*[2]

[1]Department of Structures, Faculty of Civil Engineering, VSB – Technical University of Ostrava, Ostrava, Czech Republic
[2]Department of Building Materials and Diagnostics of Structures, Faculty of Civil Engineering, VSB – Technical University of Ostrava, Ostrava, Czech Republic

david.mikolasek@vsb.cz, antonin.lokaj@vsb.cz, kristyna.vavrusova@vsb.cz, david.bujdos@vsb.cz

Abstract. *In construction practice, it is often necessary to change the use of the original building in the existing building - hall. Associated with this is the basic question of whether it is possible and under what conditions to adapt this original building to the new demands on the operating load and its change. In this case, it is a general change of use of existing floors made of wire concrete or plain concrete. Normally, these floors are primarily designed for surface loads taking into account the small point force. Due to the trend of using older buildings for new purposes (storage halls with racks - point load from rack assemblies or as in this case a built-in hall for clean rooms) it is appropriate to find a calculation and testing tool to determine the maximum load capacity of such floors. One possibility is numerical analysis using advanced modeling methods with the support of diagnostic methods and laboratory experiments to provide input data for the mathematical model. This article is a numerical analysis supplemented by knowledge from the literature and physical measurements.*

Keywords

Compressive strength, crack, concrete, FEM analysis, EPS100, FRC, stiffness and deformation, soil analysis.

This work presents the results of a numerical model of the effect of subsoil stiffness and heel plate stiffness on the change in stress in polystyrene and wire concrete (see Fig. 1).

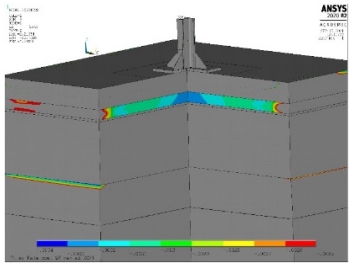


Fig. 1: Stress distribution under the contact plate.

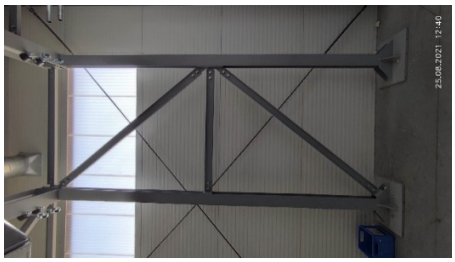


Fig. 2: Example of real anchoring steel column on slabs.

The motivation for this analysis is the long-term cooperation with the company Block, which deals with clean spaces. This company often uses older buildings and builds new halls into them, where it assembles closed halls for special requirements for dust-free. For the original buildings, the existing floors are not dimensioned for the new point load and therefore this analysis is performed here. The numerical calculation includes three types of base plates and three types of subsoil. These individual input data are combined with each other to determine the effect of these parameters on the stress in concrete and polystyrene. The result of the analysis are graphs of the course of deformations depending on the input parameters of the model (stiffness of the base plate and stiffness of the subsoil). The aim is to determine the load-bearing capacity of polystyrene and wire-reinforced concrete, so that there is no permanent deformation of polystyrene and unsatisfactory cracks in wire-reinforced concrete, which affect the stiffness of the board and thus the local pressure on EPS100.

Acknowledgements

Financial support from VSB-Technical University of Ostrava Department of Structures and for data provided by Block a.s. based in Valašské Meziříčí. Furthermore, thanks are given to the Department of Building Materials and Diagnostics of Structures for service and professional assistance in obtaining the stiffness and strength characteristics of the underlying polystyrene EPS100.

COMPARISON OF APPROACHES TO RELIABILITY VERIFICATION OF EXISTING STEEL RAILWAY BRIDGES

Vitali NADOLSKI, Milan HOLICKY, Miroslav SYKORA

Department of Structural Reliability, Klokner Institute, Czech Technical University in Prague, Prague, Czech Republic

Vitali.Nadolski@cvut.cz, Milan.Holicky@cvut.cz, Miroslav.Sykora@cvut.cz

1. Introduction

Many existing steel railway bridges are exposed to degradation and increasing traffic loads. Their reliability assessment may contribute to sustainability in construction particularly when these bridges are kept in service. The sustainability aspects mainly relate to reducing waste material and to unnecessary replacements of existing bridges. The key aspects of assessment include specification of the target reliability and selection of an appropriate assessment method. The submitted study is aimed at improvements methods of reliability assessment of an existing steel railway bridge.

2. Comparison of Approaches to Reliability Verification

Reliability requirements following from the fixed partial factors (*FPF*) provided in EN 1990 and SZ S5, adjusted partial factors (*APF*) according to SZ S5, and probabilistic method (*PM*) are critically compared. An existing steel railway bridge with age 90 years and remaining service life 10 years is considered.

Figure 1 displays variation of W_i / W_{EN} with load ratio χ for the case where a rail traffic load effect is based on the LM71 model. W_i denotes the geometrical characteristic of a cross-section (such as section module) required to satisfy the limit state in accordance with a particular approach to reliability verification and W_{EN} is the reference value based on the partial factors recommended in Eurocodes.

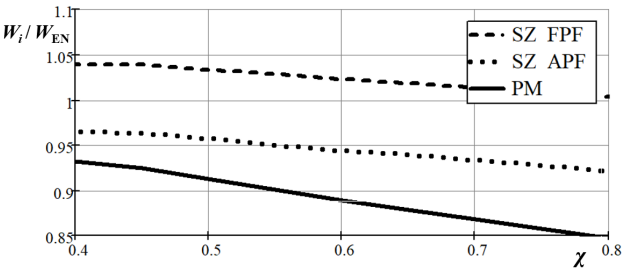


Fig. 1: Variation of W_i / W_{EN} with χ for LM71 model (age of bridge – 90 years, remaining service life – 10 years)

Figure 2 displays variation of the FORM sensitivity factors with the load ratio χ. In this case, the α-factors are practically independent of the rail traffic load model.

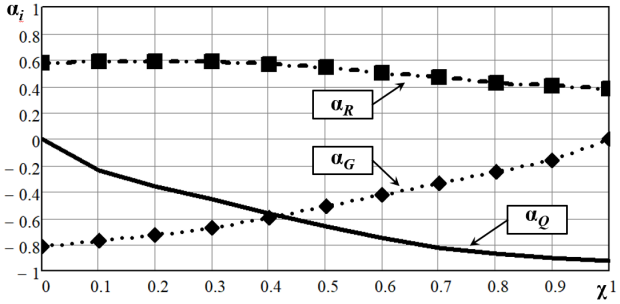


Fig. 2: Variation of sensitivity factors with χ

It appears that for steel bridges, the dominant influence on reliability can be attributed to variability of the rail traffic load effect. Detailed numerical studies indicate that in this case, the sensitivity factors depend insignificantly on a reference period as they are primarily affected by uncertainties in time-invariant variables including the dynamic amplification factor.

3. Conclusions

Application of probabilistic-based partial factors and adjusted partial factors allows to reduce assessment requirements by 10-15%. Development and wider use of the adjusted partial factors seems to be reasonable considering the balance between demands on input information, computational complexity, and achieved improvements in reliability assessments. For routine applications in assessments, the target reliability levels for existing railway bridges should be specified.

4. Acknowledgements

This study has been supported by the Ministry of Culture of the Czech Republic under Grant DG18P02OVV033.

ASSESSMENT OF THE POSSIBILITY OF USING THE TENSEGRITY SYSTEM IN THE STEEL TOWER STRUCTURE

Przemysław PALACZ[1], *Izabela MAJOR*[2]

[1] Czestochowa University of Technology, Faculty of Civil Engineering, Akademicka 3 Street, 42-200 Czestochowa, Poland

[2] Czestochowa University of Technology, Faculty of Civil Engineering, Akademicka 3 Street, 42-200 Czestochowa, Poland

przemyslaw.palacz@pcz.pl, izabela.major@pcz.pl

Abstract. *Tensegrity structures, due to their lightness and high stiffness, have been popular among architects and constructors for many years. When designing building and engineering structures, the most important stage of designing is the selection of the appropriate structural system depending on the purpose and location of the designed facility. When selecting the load-bearing structure, designers are guided by the smallest possible weight of the structure, while ensuring the safe operation of the facility, meeting all strength conditions. The article presents an analysis of the effectiveness of using the tensegrity structure for the load-bearing structure of a tower in a steel structure. The assessment of effectiveness was carried out taking into account the economy of execution and the fulfillment of the ultimate and serviceability limit conditions of individual elements of the structure for the load-bearing system of the tensegrity structure. Numerical calculations and cross-section dimensioning were performed using the finite element method in RFEM, modeling the structure in 3D.*

Keywords

Tensegrity structure, tension structure, RFEM.

1. Computational model of tower structure in the tensegrity system

For the analysis, the construction of a tower consisting of simplex tensegrity modules was adopted, with the base described in the figure of a triangle, inscribed in a circle with a diameter of 3 m. Fig. 1 shows two types of connections in multi-module tensegrity systems. The height of the tower was assumed to be 12 m, and the height of one module was assumed to be 3 m. The upper base of the module in relation to the lower base was turned by 150°. The tower consists of 5 modules overlapping each other by a distance of 1/4 of the module height. The tensegrity structure uses the opposite direction of twisting the base of the module compared to the previous one. The model was modeled in RFEM and calculations were made.

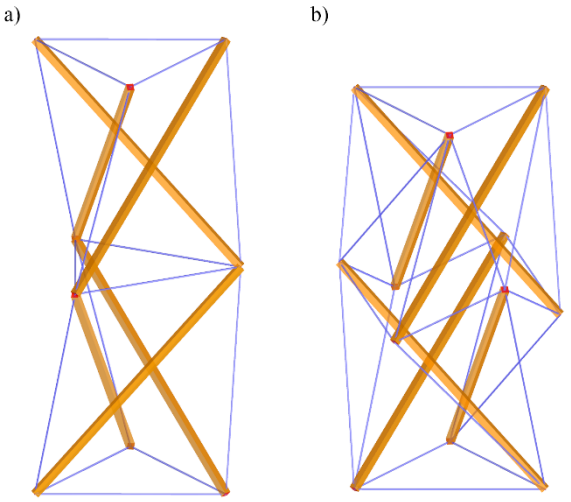


Fig. 1: Types of connection of tensegrity modules: a) node-node connection, b) strut-tendon connection.

2. Conclusion

The article presents the tensegrity structure as an interesting solution in the mechanics of structures in engineering facilities. The uniqueness of such a system is determined by the self-stress state and the infinitesimal mechanisms that stiffen it. Due to the axial distribution of forces in individual elements of the structure, some of them can be replaced with light cables, which significantly contributes to reducing the weight of the structure. It is worth noting, however, that in the tensegrity structure there is a risk of failure of the entire structure as a result of damage or tearing of one of the elements, and this may lead to the destruction of the entire structure and is disadvantageous from the point of view of the reliability of the structure. However, as a result of the analysis carried out on the basis of the modeled, loaded and dimensioned tower model, it can be concluded that the tensegrity structure for the tower structure is an interesting solution that is worth considering when designing.

APPLICATION OF SEMI-PROBABILISTIC METHODS TO VERIFICATION OF SERIES SYSTEM

Miroslav SÝKORA, Jana MARKOVÁ

Department of Structural Reliability, Klokner Institute, Czech Technical University in Prague, Šolínova 7, Prague, Czech Republic

miroslav.sykora@cvut.cz; jana.markova@cvut.cz

1. Introduction

Non-linear finite element analysis has become a widely used tool in reliability verification of reinforced concrete (RC) structures. However, such assessment is a challenging task due to growing complexity of structural models, large number of random variables, and small failure probabilities. Semi-probabilistic methods such as the partial factor method *PFM* or the method of Estimation of Coefficient of Variation *ECoV* are devised to yield adequate estimates in most situations. Previous studies showed that particularly the verification of series systems presents a challenge concerning applications of *PFM* and *ECoV*. Basic cases of series systems are thus analysed here; design resistances (R_d) estimated by these methods are compared with the probabilistic approach.

2. Lognormal Resistance (*Case 1*)

Case 1 is focused on a series system whose resistance R is obtained as a minimum of concrete and steel reinforcement contributions described by lognormal distributions. Fig. 1 displays variability of $R_{d,semi\text{-}probab} / R_{d,probab}$ with reinforcement ratio ρ. It appears that *PFM* provides a very good approximation while *ECoV* may overestimate up to 15%.

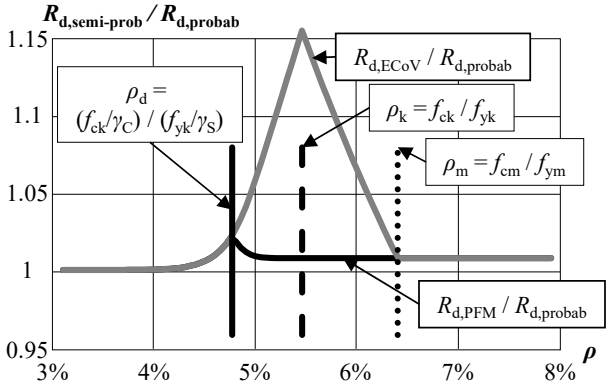


Fig. 1: Variability of $R_{d,semi\text{-}probab} / R_{d,probab}$ with ρ (*Case 1*).

3. Normal Resistance (*Case 2*)

Case 2 investigates the same series system as in *Case 1*, but concrete and yield strengths are now described by a normal distribution. Variation of the R_d-ratios is similar to that observed in *Case 1*. However, the *ECoV* error is magnified, reaching up to 30% for $R_{d,ECoV}(\rho_k)$ and converging to 10% for high reinforcement ratios.

4. Concluding Remarks

It is shown that for the series systems, *PFM* provides very good approximations. *ECoV* may overestimate in some cases; the maximum error is attributed mainly to failure in identifying a type of distribution of system resistance. When the normal distributions of component resistances are considered, the error of *ECoV*—strictly based on the assumption of lognormally distributed resistance—is magnified. Further research should be focused on the analysis of parallel and mixed series-parallel systems, cases with strongly non-lognormal resistance, effects of many components of the system and correlations between component resistances, and performance of advanced methods for reliability verification of RC structures.

Acknowledgements

This study has been supported by the Czech Science Foundation under Grant 20-01781S.

INVESTIGATION OF NEGATIVE SHEAR LAG EFFECT IN CONTINUOUS DOUBLE T BEAM BY 1D FEM

Dang-Bao TRAN[1,2], *Jaroslav NAVRÁTIL*[1]

[1] Department of Structures, Faculty of Civil Engineering, VSB–Technical University of Ostrava, Ludvíka Podéště 1875/17, Ostrava, Czech Republic.
[2] Department of Civil Engineering, Faculty of Architecture, Thu Dau Mot University, Tran Van On 06, Binh Duong Province, Vietnam.

dang.bao.tran@vsb.cz, jaroslav.navratil@vsb.cz.

Abstract. *This paper presents the use of a finite element method (FEM) to analyse the negative shear lag effect on prestressed continuous double T beam. However, the numerical method can be applied to i) any cross-section, ii). the most common types of supports, such as fixed, pinned, roller, iii) and any applied load, concentrated, or distributed. The characteristics of the cross-section are firstly derived from 2D FEM, which uses a 9-node isoparametric element. Then, a 1D FEM, which uses a linear isoparametric element, is developed to compute the deflection, rotation angle, bending warping parameter, and stress resultants. Finally, the stress field is obtained from the local analysis on the 2D-cross section. A MATLAB program is executed to validate the numerical method.*

Keywords

Shear lag, restrained warping, prestressing, load, finite element method

1. Introduction

According to the elementary beam theory, when the beam element is under load, the longitudinal normal stresses are assumed to be proportional to the distance from the neutral axis. However, in practice, these stresses are nonuniformly distributed over the width of the cross-section. This phenomenon is called shear lag. Many authors have researched the shear lag phenomenon. Most studies focus on concentrated and uniformly distributed loads. The effect of prestressed load on shear lag has only been performed in a few studies. Chang, S.T. (2004) used an analytical method to calculate the shear lag of simply supported prestressed concrete. Zhou, S.J. Qin. (2011) proposed a new FEM to analyse the shear lag in prestressed concrete box girders.The purpose of this paper is to investigate the negative shear lag effect due to prestressing load in continuous double T beam by establishing a numerical method using FEM, based on displacement and strain fields derived from Dikaros, I.C. et al. (2014).

2. Validation Example

A prestressed continuous beam with a double T cross-section shown in Fig. 1 was analysed using the code developed in MATLAB R2015. Six straight tendons are positioned uniformly at the centre of the slab, each of which is stressed with a force of 140 kN. Modulus of elasticity E= 30 x 10^3 MPa, and the Poisson ratio $\nu = 0.2$.

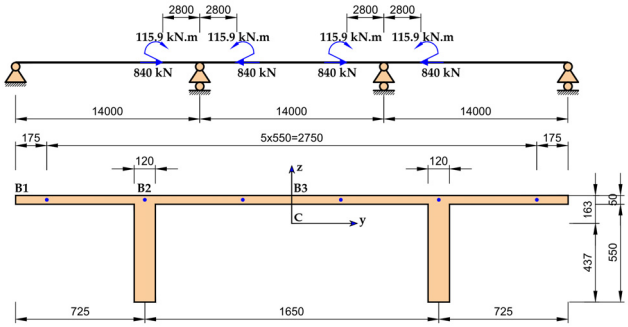


Fig. 1: Continuous beam, the load, and the geometry of the cross-section, units [mm].

3. Conclusions

In this paper, FEM is developed to analyse the negative shear lag effect due to prestressing in continuous double T beam. However, the numerical method can be used for arbitrary cross-sections with most boundary conditions and load types. A three-span continuous beam subjected to the prestressing load at the internal support was analysed and compared with the results from 3D simulation software. It was observed that the present study could predict the negative shear lag phenomenon accurately due to prestressing load.

LINEAR ANALYSIS OF NONUNIFORM TORSION WITHOUT SHEAR DEFORMATION EFFECT USING FEM

Dang-Bao TRAN[1,2], *Jaroslav NAVRÁTIL*[1]

[1] Department of Structures, Faculty of Civil Engineering, VSB–Technical University of Ostrava, Ludvíka Podéště 1875/17, Ostrava, Czech Republic.
[2] Department of Civil Engineering, Faculty of Architecture, Thu Dau Mot University, Tran Van On 06, Binh Duong Province, Vietnam.

dang.bao.tran@vsb.cz, jaroslav.navratil@vsb.cz.

Abstract. *This paper presents the use of a finite element method (FEM) to analyze nonuniform torsion with an arbitrary cross-section with homogeneous elastic material without shear deformation effect. Beams are constrained by the most common types of supports, such as fixed, pinned, and roller, and are subjected to any applied torsional load, concentrated, and distributed. The presented FEM transforms the 3D analysis of nonuniform torsion beams into separated 2D cross-sectional and 1D modelling. The geometric constants of the cross-section are firstly derived from 2D FEM, which uses a 9-node isoparametric element. Then, a 1D FEM, which uses the Hermitian shape function, is developed for computing the twist angle, the derivative of twist angle, and stress resultants. Finally, the stress field is obtained from the local analysis on the 2D-cross section. A MATLAB program is executed to validate the numerical method through examples. The validation examples have proven the reliability of the author's numerical method for analyzing problems defined above.*

Keywords

Nonuniform torsion, Shear deformation effect, Warping restraint.

1. Introduction

In engineering practice, we often encounter beam structures loaded in torsion. When the warping of a member's cross-section is not restrained, the stress field of a prismatic beam with homogeneous isotropic material can be derived from the Saint-Venant theory strictly [1-4]. In practice, because i) boundary conditions are imposed, ii) geometrical characteristics of the section, the warping of beam's cross-section is restrained, which leads to additional normal and shear stresses, which the Saint Venant theory does not take into account. Vlasov [5] was the first to formulate the problem of nonuniform torsion. Benscoter [6] improved Vlasov's theory, which neglects the shear deformation effect, leading to errors with closed cross-sections [7]. Many authors established FEM to consider nonuniform torsion taking into account the shear deformation effect [7-14]. However, the above research [7-14] used the approximations of Thin Tube Theory [5] to determine the bar's geometric constants, restricting the accuracy and applicability of the formulations. The purpose of this paper is to establish a numerical method using FEM to solve the nonuniform torsion problem without the shear deformation effect of the prismatic beam with arbitrary cross-section unchanged throughout the length with homogeneous isotropic elastic material, based on displacement and strain fields derived from Sapountzakis, EJ et al. [17]. This paper is a preliminary research step for future research, which considers the shear deformation effect in nonuniform torsion.

2. Validation Examples

A computer code is developed in the MATLAB R2015a software based on the formulations described in the previous sections. The obtained results are compared with the available works of the literature.

3. Conclusion

In this paper, a FEM is developed to solve nonuniform torsion without the shear deformation effect of the prismatic beam with an arbitrary cross-section with homogeneous isotropic material. Two examples were performed for validating the accuracy of the present study. The comparison of the results from the present study with the corresponding results in the literature shows that the present study can predict the responses of the non-uniform torsion with arbitrary cross-section without shear deformation effect accurately. Linear analysis of nonuniform torsion considering shear deformation would be one of the future research topics.

MOŽNOST PROVEDENÍ DODATEČNÝCH OTVORŮ V NOSNÝCH STĚNÁCH STÁVAJÍCÍ PANELOVÉ VÝSTAVBY

Miroslav VACEK[1], *Miroslav ROSMANIT*[1], *David MIKOLÁŠEK*[1]

[1]VŠB – Technická univerzita Ostrava, Fakulta stavební, Ludvíka Podéšte 1875/17, Ostrava - Poruba

miroslav.vacek@vsb.cz, miroslav.rosmanit@vsb.cz, david.mikolasek@vsb.cz

Abstract. *A frequent requirement for the innovation of housing units in residential buildings made of prefabricated concrete panels is the creation of a new opening in the load-bearing wall of the building. The article discusses one selected reconstruction case leading to a design of supporting steel frame structure activated before the start of demolition work.*

Klíčová slova

Panelové domy, otvor, ocelový výztužný rám.

1. Úvod

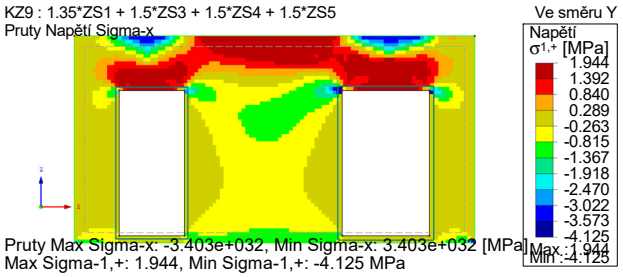
Rekonstrukce bytových jader v panelových domech je dlouhodobě řešené téma stavební praxe. S rekonstrukcí se hojně váže problém s propojením dvou sousedících místností novým otvorem budovaným ve stávající nosné stěně panelového domu. Velkou pozornost, při řešení podobného problému, je potřeba věnovat především vzniku trhlin, které se však, dle dostupných poznatků, vyskytují převážně v průběžných spárách panelů [1]. Vznik trhliny v ploše panelu již před započetím bourání otvoru, značí globální problém celého nosného systému a pravděpodobné přetížení části konstrukce. Budování jakýchkoliv nových otvorů u těchto staveb je nemožné.

V případě dobré kondice nosného systému objektu je vzniku nových otvorů možný, doporučuje se však jeho šířkové omezení do max. 1000 mm. Zároveň je nutné ověřit, zda v okolních panelech již není dodatečný otvor zbudován. Pokud v sousedních, především nad horním a pod dolním lícem řešeného panelu panelech, již otvor v minulosti zbudován byl, je nutné ověřit také vliv těchto otvorů na plánovaný zásah do posuzované konstrukce [2].

V dostupné literatuře jsou uváděny mnohé limity pro provádění a doporučení na podepření konstrukce při vyřezání otvoru. Tomuto se snaží autoři článku vyhnout navržením předem aktivovaného ocelového průvlaku, který bude ještě dále vyztužen po vyřezání otvoru. Navržený postup prací je v článku podrobně popsán.

Pro analýzu chování konstrukce při budování nového otvoru byl vybrán modifikovaný panelový dům G-OS se známým zaměření stávajícího stavu a informací o počtu, velikostech a umístění dalších otvorů v nosných stěnách.

V článku je postupně osvětlena myšlenka provedení nově budovaného otvoru bez využití dodatečného podepření konstrukce. Dále je provedena analýza změny tuhosti oslabené stěny ve vodorovném směru, pro ověření roznosu zatížení větrem na oslabeném objektu. Na zjednodušeném stěnovém problému s uvážením působících svislých zatížení je provedena analýza možných řešení předem vyztuženého nadpraží. Následně je na vybraném řešení je provedena detailní numerická analýza využívající objemové konečné prvky pro nalezení kritických míst, ve kterých by mohla vzniknout lokálního porucha, která by později ovlivnila globální chování konstrukce. Autoři nabízí také možnost zjednodušeného posouzení konstrukce. Výsledky jednotlivých řešení navzájem porovnávají a diskutují. V závěru článku je prokázána proveditelnost navrženého řešení vyztužení dodatečně budovaného otvoru využitím spolupůsobící ocelové rámové konstrukce.



Obr. 1 Příklad napětí σ_1 v betonu v případě nevyztuženého otvoru

[1] WITZANY, J. a kol.: Metodické a technické pokyny pro posuzování stavebních úprav a zásahů do nosné konstrukce panelových domů, certifikovaná metodika, Praha: ČVUT v Praze, VUT v Brně, VŠB, 2016, Dostupné na: http://www.sfrb.cz/fileadmin/user_upload/metodika_otvory_final.pdf

[2] WITZANY, Jiří, Jaromír VRBA a Václav HONZÍK. Otvory v panelových domech. Praha: Pro Českou komoru autorizovaných inženýrů a techniků činných ve výstavbě vydalo Informační centrum ČKAIT, 2014. Technická knižnice (ČKAIT). ISBN 978-80-87438-55-8.

THE IMPORTANCE OF MESH MODULE IN FEM ANALYSIS OF PUSH-OUT TESTS

Patrícia VAŇOVÁ[1], *Vincent KVOČÁK*[1]

[1]Institute of Structural Engineering, Faculty of Civil Engineering, Technical University of Košice, Vysokoškolská 4, 042 00 Košice, Slovakia

patricia.vanova@tuke.sk, vincent.kvocak@tuke.sk

Abstract. *Standard push-out tests are nowadays often performed by many researchers around the world in order to determine the elastic shear resistance of the selected shear element. Later, these experiments are modelled in several softwares based on finite element method (FEM). The main toping of this article is the importance of choosing the right mesh settings for such analysis – shape, curvature control and specifically size. Their effect is closely described below.*

Keywords

Mesh, FEM, push-out tests, steel-concrete, continuous shear connector.

1. Introduction

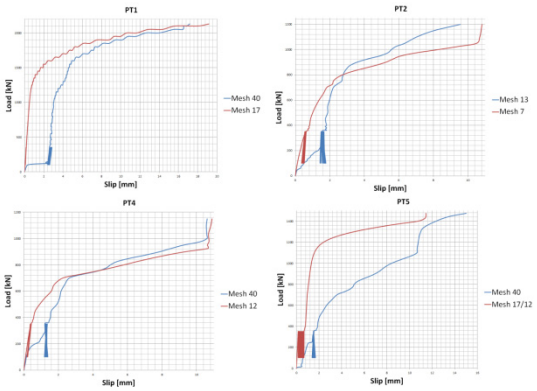


Fig. 1: Comparison of results between bigger and lower global size of each push-out test analysis.

2. Conclusion

From the Fig. 5 it is visible how big effect the size of the elements has onto the analysis results. This is also specific to the push-out tests due to the small distance – 10 mm – of the measured slip. In all the compared analysis the difference in results occurred before the cycle between 40 and 5% of the expected resistance, with deviation lower than 1mm. In the last type of the strip, PT5, the mesh size of concrete and steel had to be divided into different numbers, probably due to the small, 20mm in diameter, holes in its walls.

Based on the trial and error method it was found that the maximum deviation control as well as minimum size control performed well when the ratio was around 0.1 for all parts and that the global size of the parts of push-out tests was acceptable when lower or equal to 20, with reinforcement at its higher limit and steel parts with lower numbers.

Acknowledgements

The paper has been supported by The projects: VEGA 1/0172/20 “Stress and deformation analysis of load bearing components made out of steel, glass and composite materials” of the Scientific Grant Agency of the Ministry of Education, science, research and sport of the Slovak Republic and the Slovak Academy of Sciences and by the project Slovak Research and Development Agency under the contract No. APVV-15-0486 “Analysis of Shear Connection Influence in Bridges with Encased Beams”.

References

[1] STN EN 1994-1-1. *Eurokód 4: Navrhovanie spriahnutých oceľovobetónových konštrukcií. časť 1-1: Všeobecné pravidlá pre budovy*, Technical report, 2006, SÚTN Bratislava.

[2] Dassault Systèmes. Abaqus 6.13 online documentation. 2013. Available at: http://dsk.ippt.pan.pl/docs/abaqus/v6.13/books/usi/default.htm

Effect of the mechanical fracture parameters of inclusion on fracture behaviour

Michal VYHLÍDAL[1]

[1]Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Veveří 331/95, 602 00 Brno, Czech Republic

vyhlidal.m@fce.vutbr.cz

Abstract. *This paper concerns the results of research into the effect of the mechanical–fracture parameters of inclusion material on the fracture response of specially designed cement-based composite specimens. These specimens of the nominal dimension* $40 \times 40 \times 160$ mm *with inclusion in the shape of prisms with nominal dimensions of* $8 \times 8 \times 40$ mm *were provided with an initial central edge notch and tested in the threee-point bending configuration, see Fig. 1. These specimens were made of fine-grained cement-based composite and had different types of inclusion – rock inclusions (amphibolite, basalt, granite and marble), steel and extruded polystyrene.*

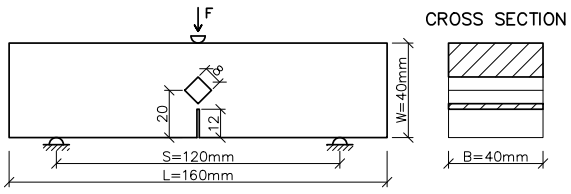


Fig. 1: Specimen geometry and the three-point bending fracture test configuration.

The aim of this paper is to analyse the effect of the mechanical–fracture parameters of inclusions material on the effective mechanical–fracture parameters of cement-based composite. For this purpose, the correlations between evaluated mechanical–fracture parameters evaluated from F–d diagrams and mechanical–fracture parameters of inclusion material are presented. Namely Young's modulus of elasticity E_{agg} and Poisson's ratio ν_{agg} of inclusion are used, other predominantly fracture parameters are omitted. The reason is simple – no crack propagation through the inclusion was observed (except of the first specimen with marble inclusion) so the fracture properties of the inclusion should not affect the fracture behaviour of specimens.

The results of this research indicate the dependence of the Young's modulus of inclusion material E_{agg} on the effective mechanical–fracture parameters of cement-based composite. The results show that Young's modulus of composite E is negatively correlated with Young's modulus of inclusion material E_{agg}. The resistance, which is approximately expressed by the values of maximal forces F/F_{mtx}, specific fracture energy $G_{\mathrm{F}}/G_{\mathrm{F,mtx}}$ and effective fracture toughness $K_{\mathrm{Ice}}/K_{\mathrm{ICe,mtx}}$ ratios, is maximal in the case of $E_{\mathrm{agg}} \approx 2E_{\mathrm{mtx}}$.

The maximal values of $G_{\mathrm{F}}/G_{\mathrm{F,mtx}}$ and $K_{\mathrm{Ice}}/K_{\mathrm{ICe,mtx}}$ are in the case of $\nu_{\mathrm{agg}} \approx 0.2$ which is the value typical for concrete. Nevertheless, the effect of the Poisson's ratio of inclusion ν_{agg} is not as significant as in the case of Young's modulus E_{agg}.

Anyway, the author is aware that it is necessary to perform more experiments and confirm it by numerical simulations to prove the real effect of mechanical–fracture parameters of inclusion on overall fracture behaviour of composite.

Keywords

Cement-based composite, Fracture test, Inclusion, Mechanical fracture parameters.

Acknowledgment

This outcome has been achieved with the financial support of the Brno University of Technology under project No. FAST-J-21-7497. The author would also like to thank many kind colleagues, namely Patrik Bayer, Petr Daněk, Barbara Kucharczyková, Iva Rozsypalová, Hana Šimonová, Zbyněk Keršner and others who lent a helping hand.

Advantages of Tensile Structure Analysis During Construction Stages

Zbyněk ZAJAC, Rostislav LANG, Ivan NĚMEC

Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Veveří 331/95, 602 00 Brno, Czech Republic

zajac.z@fce.vutbr.cz, lang.r@fce.vutbr.cz, nemec.i@fce.vutbr.cz

Abstract. *This paper deals with the importance of analysing tensile structures during individual construction stages. This type of analysis is usually related to conventional structures. However, results of the research on this topic imply that problems caused by an insufficient design are solved on a building site on a daily basis. It usually is the engineer's responsibility to find a proper solution and a method of construction. A challenging decision is strongly dependent on experience and current status. This research contributes to a more detailed description of the problem.*

Keywords

Construction stages, form-finding, tensile surface structures.

1. Introduction

This paper focuses on the importance of construction stages analysis of tensile surface structures. Lightweight structures represent a sustainable and elegant roofing solution. In order to reach sufficient efficiency of the individual elements it is important to include each response in the calculation.

Membrane structures are characterized by internal membrane forces due to the fact that the bending stiffness of a membrane material is almost zero. The structural shape of a tensile structure is determined by forces, thus from an inverse formulation of equilibrium. [?]

In general, the spatial position of all material points of the surface are determined in order to reach equilibrium during the form-finding procedure. Input parameters are external forces and internal stresses acting on the unknown shape of the surface. [?]

2. Conclusion

Tensile surface structures are very sensitive to any change as has been stated and proved. A temporary load may affect the structure under construction in an unexpected way during the most vulnerable condition. This approach for analyzing the membrane surface structures may help to suggest new construction methods. Load distribution appears to be one of the most important objectives. A more uniform distribution of the principal internal forces n_1 ensures a better initial condition for further loading. Another important objective is to reduce the overall amount of extreme internal forces. Then it is possible to choose smaller steel parts of the surface. Thus, the design of the structure is more effective and ecological.

Acknowledgment

This paper has been created with the financial support of the specific university research project granted by Brno University of Technology, registered under the number FAST-J-21-7496.

References

[1] BLETZINGER, K., R. WÜCHNER and F. DAOUD and N. CAMPRUBÍ. Computational methods for form finding and optimization of shells and membranes. *Computer Methods in Applied Mechanics and Engineering*. 2005, vol. 194, iss. 30-33, pp. 3438-3452. ISSN 0045-7825. DOI: 10.1016/j.cma.2004.12.026.

[2] LINHARD, J., K. BLETZINGER. "Tracing" the equilibrium - Recent advances in numerical form finding. *International Journal of Space Structures*. 2010, vol. 25, iss. 2, pp. 107-116. ISSN 0956-0599. DOI: 10.1260/0266-3511.25.2.107.

DAMAGES DIAGNOSIS IN MASONRY ELEMENTS BY NUMERICAL MEASURES OF VIBRATION PROCESS

Mariusz ŻÓŁTOWSKI[1], *Olga SZLACHETKA*[2]

[1]Institute of Civil Engineering, Faculty of Civil and Environmental Engineering, Warsaw University of Life Sciences – SGGW, Nowoursynowska Street 166, Warsaw, Poland.
[2]Institute of Civil Engineering, SGGW Water Centrum, Warsaw University of Life Sciences – SGGW, Nowoursynowska Street 166, Warsaw, Poland.

mariusz_zoltowski@sggw.edu.pl, olga_szlachetka@sggw.edu.pl.

Abstract. *The aim of the study was to test the effectiveness of the proprietary FUNCTIONS ANALYSIS software to assess the degradation of tested brick elements based on the surface areas of the obtained FRF functions. The research confirmed the usefulness of experimental modal analysis in identifying damages of masonry elements based on a comparison of the course of the FRF function and the coherence function in a damaged and undamaged object.*

Keywords

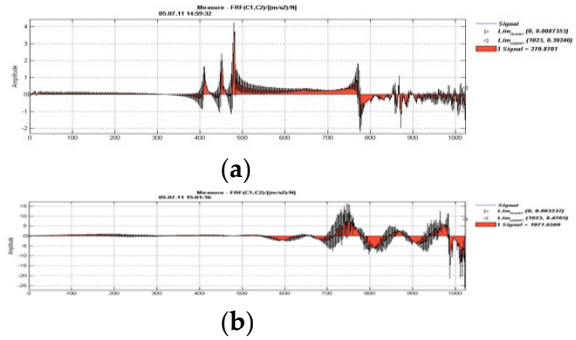
Modal analysis; vibration frequencies; stabilization diagram; solid brick; checker brick.

1. Introduction

The purpose of the work is to conduct an experimental modal analysis by the SISO method and confirm its usefulness in identifying damages of masonry elements based on a comparison of the FRF function and the coherence function in a damaged and undamaged object.

2. Validation Example

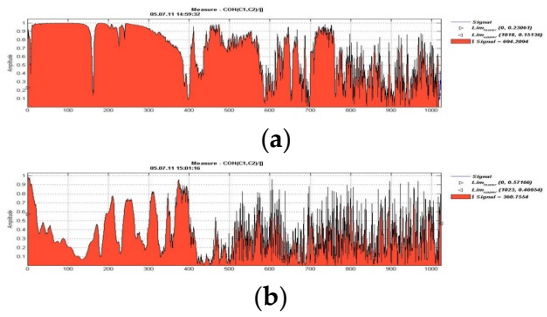
The following figures show a comparison of the average of 10 samples (undamaged and damaged), FRF function waveforms and coherence in terms of their visual time course, as well as the calculated surface areas of the function data for fit and damaged samples of solid bricks and checker bricks.



(**a**)

(**b**)

Fig. 8: The average surface area of the FRF function for solid brick (**a**) undamaged, (**b**) damaged in the Z axis.



(**a**)

(**b**)

Fig. 9: Average surface area of the coherence function for solid brick (**a**) undamaged; (**b**) damaged in the Z axis.

3. Conclusion

Presented research results indicate that basically a modal analysis can distinguish between material properties, which affects the ability to distinguish between their strength properties or degradation state. The research confirmed the usefulness of experimental modal analysis in identifying damages of masonry elements based on a comparison of the course of the FRF function and the coherence function in a damaged and undamaged object.

VSB TECHNICAL UNIVERSITY OF OSTRAVA | FACULTY OF CIVIL ENGINEERING

VSB-Technical University of Ostrava, Faculty of Civil Engineering
Vysoká škola báňská – Technická univerzita Ostrava, Fakulta stavební

19th International conference / 19. mezinárodní konference

MODELLING IN MECHANICS 2021 / MODELOVÁNÍ V MECHANICE 2021

21. - 22. 10. 2021

Proceedings of extended abstracts / Sborník rozšířených abstraktů

Topics / Tematické okruhy

The conference is focused on the following topics /
Konference je zaměřena na následující tematické okruhy:

- development and application of numerical methods in mechanics / rozvoj a aplikace numerických metod v mechanice,
- methods used in extensive tasks dealing with mechanics of continuum / metody řešení rozsáhlých úloh mechaniky kontinua,
- numerical modelling of static and dynamic behaviours of concrete, brick, steel, timber and composite building structures / numerické modelování statického a dynamického chování betonových, zděných, ocelových, dřevěných a kompozitních stavebních konstrukcí,
- interaction between subsoil and building structures / interakce stavebních konstrukcí s podložím,
- influence of undermining on building structures / vliv poddolování na stavební objekty,
- loads and responses of structures in extreme conditions / zatížení a odezva konstrukcí v extrémních podmínkách,
- rehabilitation, reconstruction and reinforcement of load-carrying structures in buildings / sanace, rekonstrukce a zesilování nosných konstrukcí staveb,
- statics and dynamics of building structures / statika a dynamika stavebních konstrukcí,
- automation of engineering tasks / automatizace inženýrských úloh,
- mechanics of materials / mechanika materiálů,
- non-linear mechanics / nelineární mechanika,
- fracture mechanics / lomová mechanika,
- experimental verification of structures / experimentální ověřování konstrukcí,
- modelling of structures subject to heat, including fire resistance / modelování teplotně namáhaných konstrukcí včetně požární odolnosti,
- reliability and probability tasks in mechanics / spolehlivostní a pravděpodobnostní úlohy v mechanice.

Scientific committee / Vědecký výbor konference

(in alphabetical order / v abecedním pořadí)

prof. Ing. Jiří Brožovský, Ph.D., VŠB - Technical University of Ostrava, Czech Republic,
prof. Ing. Radim Čajka, CSc., VŠB - Technical University of Ostrava, Czech Republic,
doc. Ing. Martin Čermák, Ph.D., VŠB - Technical University of Ostrava, Czech Republic,
Ing. Michal Drahorád, Ph.D., Czech Technical University in Prague, Czech Republic,
prof. Ing. Drdácký Miloš, DrSc., dr. h. c., Institute of Theoretical and Applied Mechanics Academy of Sciences of the Czech Republic,
doc. Ing. Petr Frantík, Ph.D., Brno University of Technology, Czech Republic,
prof. Ing. Milan Holický, DrSc., Czech Technical University in Prague, Czech Republic,
prof. Ing. Norbert Jendželovský, Ph.D., Slovak University of Technology in Bratislava, Slovak Republic,
prof. Ing. Petr Kabele, Ph.D., Czech Technical University in Prague, Czech Republic,
prof. Ing. Jiří Kala, Ph.D., Brno University of Technology, Czech Republic,
prof. Ing. Zdeněk Kala, Ph.D., Brno University of Technology, Czech Republic,
dr hab. inż. Jacek Katzer, prof. UWM, Center of Civil Engineering, Faculty of Geoengineering, University of Warmia and Mazury in Olsztyn, Poland,
prof. Ing. Zbyněk Keršner, CSc., Brno University of Technology, Czech Republic,
doc. Ing. Jan Klusák, Ph.D., Institute of Physics of Material Academy of Sciences of the Czech Republic,
doc. Ing. Petr Konečný, Ph.D., VŠB - Technical University of Ostrava, Czech Republic,
prof. Ing. Eva Kormaníková, Ph.D., Technical University of Košice, Slovak Republic,
doc. Ing. Kamila Kotrasová, Ph.D., Technical University of Košice, Slovak Republic,
prof. Ing. Juraj Králik, Ph.D., Slovak University of Technology in Bratislava, Slovak Republic,
prof. Ing. Martin Krejsa, Ph.D., VŠB - Technical University of Ostrava, Czech Republic,
doc. Ing. Vít Křivý, Ph.D., VŠB - Technical University of Ostrava, Czech Republic,
doc. Ing. David Lehký, Ph.D., Brno University of Technology, Czech Republic,
prof. Ing. Antonín Lokaj, Ph.D., VŠB - Technical University of Ostrava, Czech Republic,
Assoc. Prof. Eng. Izabela Major, PhD., Czestochowa University of Technology, Poland,
Assoc. Prof. Eng. Maciej Major, PhD., Czestochowa University of Technology, Poland,
doc. Ing. Jana Marková, Ph.D., Czech Technical University in Prague, Czech Republic,
prof. Ing. Alois Materna, CSc. MBA, VŠB - Technical University of Ostrava, Czech Republic and Czech Chamber of Authorized Engineers and Technicians in Construction,
prof. Ing. Jozef Melcer, DrSc., University of Žilina, Slovak Republic,
prof. Ing. Milan Moravčík, CSc., University of Žilina, Slovak Republic,
doc. Ing. Jaroslav Navrátil, CSc., VŠB - Technical University of Ostrava, Czech Republic,
doc. Ing. Ivan Němec, CSc., Brno University of Technology and FEM consulting, Czech Republic,
prof. Ing. Drahomír Novák, DrSc., Brno University of Technology, Czech Republic,
Assoc. Prof. Eng. Tomasz Ponikiewski, PhD., Silesian University of Technology, Gliwice, Poland,
doc. Ing. Stanislav Pospíšil, Ph.D., Institute of Theoretical and Applied Mechanics Academy of Sciences of the Czech Republic and VŠB - Technical University of Ostrava, Czech Republic,
doc. Ing. Martin Psotný, PhD., Slovak University of Technology in Bratislava, Slovak Republic,
doc. Ing. Stanislav Seitl, Ph.D., Brno University of Technology and Institute of Physics of Material Academy of Sciences of the Czech Republic,
doc. Ing. Miroslav Sýkora, Ph.D., Czech Technical University in Prague, Czech Republic,
prof. Ing. Jiří Šejnoha, DrSc., Czech Technical University in Prague, Czech Republic,
prof. Ing. Břetislav Teplý, CSc., Brno University of Technology, Czech Republic,
doc. Ing. Katarína Tvrdá, PhD., Slovak University of Technology in Bratislava, Slovak Republic,
prof. Ing. Josef Vičan, CSc., University of Žilina, Slovak Republic,
prof. Ing. Miroslav Vořechovský, Ph.D., Brno University of Technology, Czech Republic.

Děkujeme partnerům Fakulty stavební VŠB-TU Ostrava.

We thank the partners of the Faculty of Civil Engineering, VSB-Technical University of Ostrava.

Title / Název:	Proceedings of extended abstracts **Modelling in Mechanics** 19th International Conference 21st and 22nd October 2021 Sborník rozšířených abstraktů **Modelování v mechanice** 19. ročník mezinárodní konference 21. - 22. 10. 2019
Author / Autor:	Team of authors / Kolektiv autorů
Place, year, edition / Místo, rok, vydání:	Ostrava, 2021, 1st edition / Ostrava, 2021, 1. vydání
Number of pages / Počet stran:	38
Published by / Vydala:	VSB-Technical University of Ostrava VSB-Technická univerzita Ostrava
Press / Tisk:	Editorial Center, VSB-Technical University of Ostrava Ediční středisko, VŠB-TU Ostrava
Number of copies / Náklad:	60

Not for sale / Neprodejné

ISBN 978-80-248-4560-9 (Print)

ISBN 978-80-248-4561-6 (Online)