Đorđe Đurđević[1], Andrijana Đurđević[1], Nina Anđelić[2], Ana Petrović[2]*

NUMERICAL-EXPERIMENTAL DETERMINATION OF STRESS AND DEFORMATION STATE IN CONNECTION LUGS

NUMERIČKO-EKSPERIMENTALNO ODREĐIVANJE NAPONSKOG I DEFORMACIJSKOG STANJA NA ELEMENTIMA ZA VEZU-UŠKAMA

Originalni naučni rad / Original scientific paper
UDK /UDC:

Rad primljen / Paper received: 9.6.2021

Adresa autora / Author's address:
[1)] Tehnikum Taurunum - College of Applied Engineering Studies, Belgrade, Serbia,
email: djdjurdjevic@tehnikum.edu.rs
[2)] University of Belgrade, Faculty of Mechanical Engineering, Belgrade, Serbia

Keywords
- connection lug
- finite element method (FEM)
- experiment
- stress-strain distribution

Ključne reči
- uška
- metod konačnih elemenata (FEM)
- eksperiment
- raspodela napona i deformacija

Abstract

The present paper describes the numerical and experimental methodology for the analysis of stress and deformation state in structural elements with geometrical discontinuities. Research is based on structural elements of the connection lug type and stress and deformation state. Numerical analysis is conducted by applying the finite element method (FEM) in a KOMIPS® and ABAQUS® software package. Experiments are performed at the Laboratory for Stress and Deformation Measurements, Faculty of Mechanical Engineering, Belgrade, using GOM equipment and ARAMIS® software application. The paper demonstrates how it is possible to anticipate the results by applying FEM. A short review of current research in the field of structural elements with geometrical discontinuities is given within the framework of the paper.

Izvod

U radu je opisana numerička i eksperimentalna metodologija za analizu naponskog i deformacijskog stanja strukturnih elemenata sa diskontinuitetima geometrije. Istraživanje se bazira na strukturnim elementima tipa uške i određivanje naponskog i deformacijskog stanja. Numerička analiza je sprovedena primenom metode konačnih elemenata u softverskom paketu KOMIPS® i ABAQUS®. Eksperimenti su izvedeni u Laboratoriji za merenje napona i deformacije na Mašinskom fakultetu u Univerziteta u Beogradu, sa opremom GOM i softverskom aplikacijom ARAMIS®. Ovaj rad objašnjava kako se mogu predvideti eksperimentalni rezultati primenom metode konačnih elemenata. U okviru rada je dat pregled istraživanja iz oblasti strukturnih elemenata sa diskontinuitetima geometrije i pregled dosadašnjih rezultata postignutih u ovoj oblasti.

INTRODUCTION

This section presents a review of hitherto conducted research and results achieved in the area of stress and deformation state of structural elements with geometrical discontinuities. It is necessary to reliably determine exploitation behaviour of structural elements by applying contemporary numerical-experimental methods. The objective of the paper is to investigate stress and deformation field at the points of geometrical discontinuities of structural elements under the action of axial loading. Theoretical and experimental investigations show that in the zones, where the loaded element contour changes abruptly, a local increase of stress occurs. Such zones are notches, holes of different shapes, points of abrupt curvature, as well as contact points between two mutually acting elements. Peterson earlier reported the results from this field, /1/. The investigation was carried out using connection lug type models loaded by axial forces, /2, 3/. Theoretical solutions for plates connected by threaded fasteners are provided by Yu and Rowlands in /4/. Similar problems with stress concentrations are discussed in /5, 6/. Experimental results (Aramis) and the results obtained by numerical methods from the above mentioned fields are reported in /7, 8/. All designed and constructed structures inevitably have a change in geometry that causes stress concentration. Current standard methods of calculation and testing cannot accurately determine and anticipate the intensity of geometrical discontinuity effects on structural deformation and stress.

ANALYSIS OF CONNECTING LUGS

Geometrical shape of analysed connection lug structural elements made of structural steel S355 is presented in Fig. 1. The analysis includes one connection lug of dimensions as shown in Table 1. Axial distance between holes of the analysed connection lug is respectively 109.5 mm. Axial loading is 20 kN.

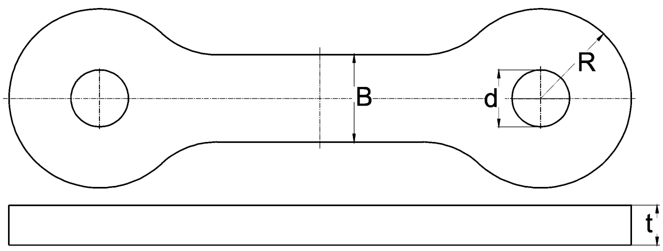

Figure 1. Geometrical shape of analysed connection lug.

Table 1. Dimensions of connection lug according to Fig. 1.

Designation of connection lug	d (mm)	R (mm)	B (mm)	t (mm)
U	14	22.5	22	10

FINITE ELEMENT METHOD

Numerical analysis is conduced by application of finite elements using KOMIPS® and ABAQUS® software, /8, 9/. Geometrical modelling and finite element mesh are given here. First the model is built in KOMIPS software in such a way as to consist of plate type and contact type elements, /10/. The contact between the axle and the connection lug is modelled via contact elements. The axle is considered as a rigid element and is used for load input. Figure 2 presents the finite element mesh of the appropriate connection lug.

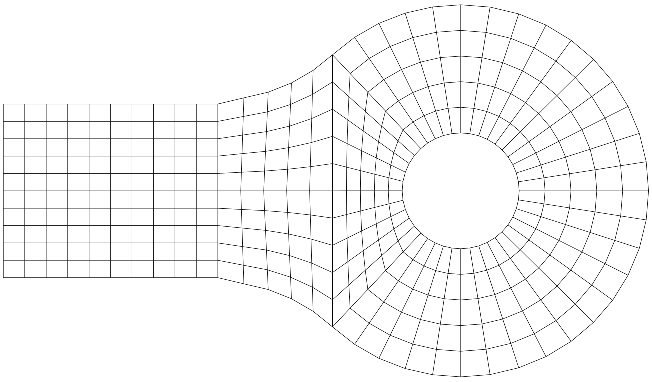

Figure 2. Finite element mesh of connection lug in KOMIPS and ABAQUS software.

EXPERIMENTAL WORK

The aim of experimental investigation is to verify the numerical calculation of the model using physical (previously created) laboratory models. The experiment was performed at the *Laboratory for stress and deformation measurements*, at the Faculty of Mechanical Engineering, University of Belgrade. A fairly new experimental method for contactless measurements, for 3D optical strain and stress analysis based on Digital Image Correlation (Aramis-system) is used, /11, 12/.

The preparation of the experiment is given in this chapter. In order to perform measurements, a stochastic sample needs to be applied to the measuring points. The stochastic pattern is formed by first applying white matte spray paint, and later applying black paint lightly from a certain distance, so that black spots are formed (Fig. 3).

Experimental setup is presented in Fig. 4. The vertical force is delivered by the hydraulic cylinder leaning against the upper horizontal beam. The aim is to keep the loading in a closed contour of the rigid frame, where two horizontal beams of the frame and vertical beams make up a closed contour. Arrangement of measuring equipment is presented in Fig. 5.

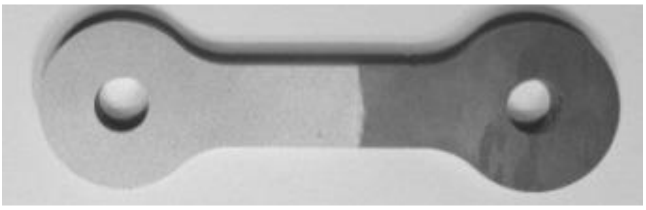

Figure 3. Stochastic pattern applied onto measuring points.

Figure 4. Measurement setup.

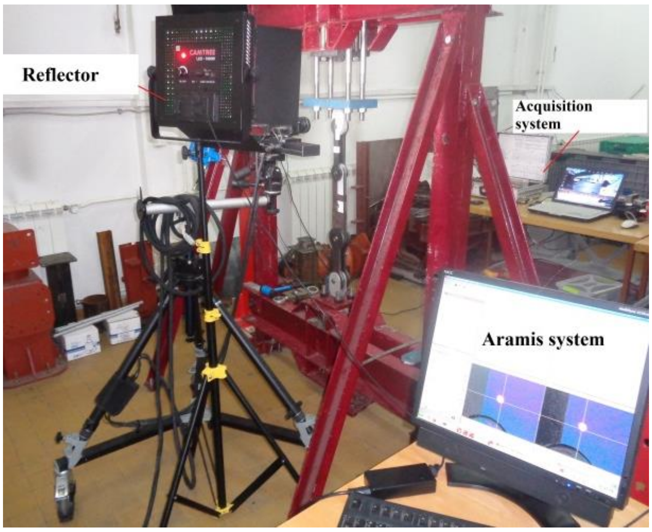

Figure 5. Arrangement of measuring equipment.

Each measurement involved a gradual input of force using a hydraulic cylinder at increments of 10 kN. The reading and control of force input is performed using a dynamometer with measuring tapes, attached to the data acquisition device (Fig. 6). Figure 7 shows the connection tab set for measurement.

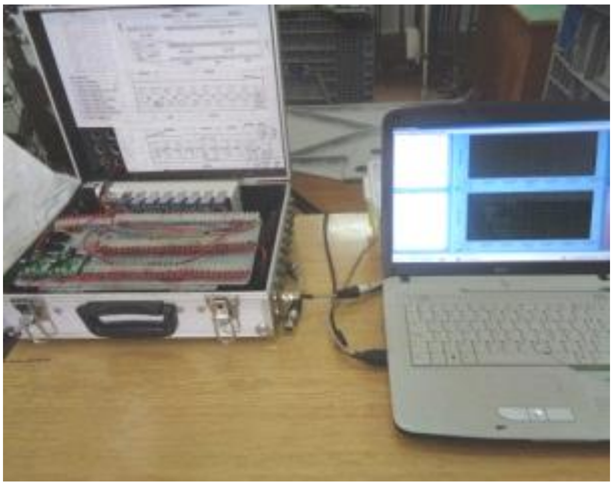

Figure 6. Data acquisition system.

Figure 7. Connection lug positioned for measurement.

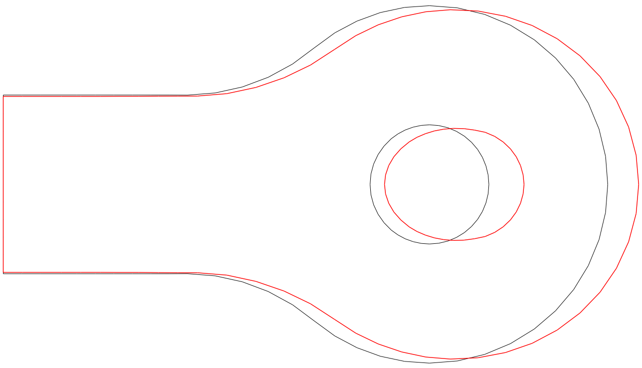

Figure 8. Deformation of the model in KOMIPS software.

RESULTS

Results obtained by KOMIPS are shown in Fig. 8 for displacements, with maximal value 0.045 mm, and in Fig. 9 for von Mises stresses (MPa). Results obtained by ABAQUS are shown in Figs. 10 and 11 for displacements, where the maximal value is 0.035 mm, and for stresses, respectively.

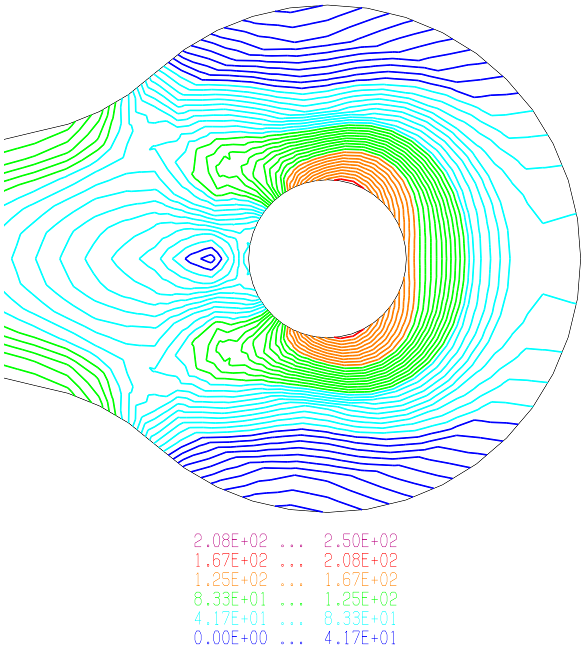

Figure 9. Von Mises stress field (σ) with associated scale for the connection lug using the KOMIPS software.

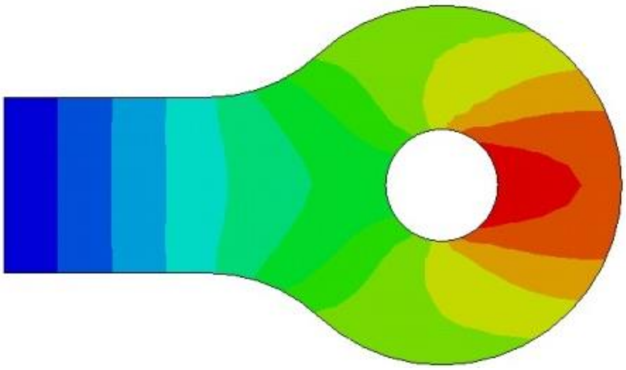

Figure 10. Deformation of the model in ABAQUS software.

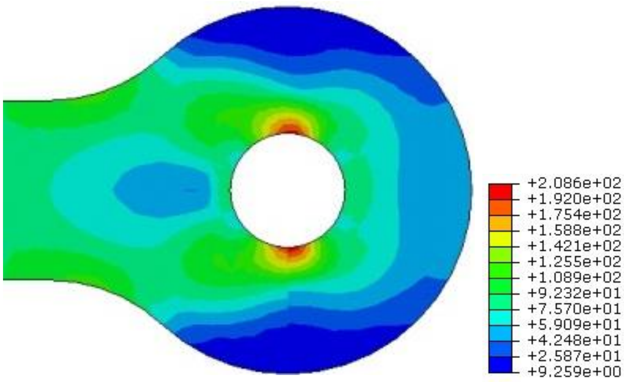

Figure 11. von Mises stress field (σ) with associated scale for connection lug using the ABAQUS software.

Figure 13 shows the measurement results obtained by Digital Image Correlation (DIC) and *Aramis* system of non-contact measurement of strain, /11/.

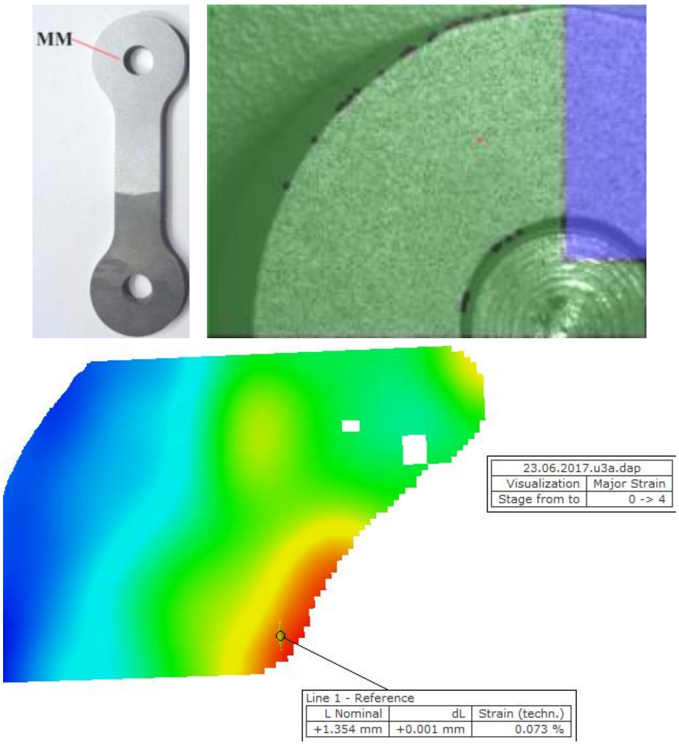

Figure 13. Experimental results for the connection lug.

DISCUSSION

Table 2 presents the results obtained by finite element method using KOMIPS and ABAQUS software, compared to experimental results. The difference in values of stresses in the results obtained numerically and experimentally is within the acceptable limits.

Table 2. Comparison of obtained stresses (MPa).

FEM analysis KOMIPS	FEM analysis ABAQUS	Experiment
158	160	155

CONCLUSIONS

The present paper is the result of many years of research on the deformation and stress state of structural elements with discontinuities in the geometry. Initial considerations are given which include a review of previous research of the stress and strain state of structural elements with discontinuities in the geometry, as well as a review of available literature. Strength diagnostic methods are considered in the preparation of this paper. These are the finite element method, as a numerical method, and the digital image correlation method, as an experimental method.

Calculation models of the structural elements have been made and a static calculation is performed using the finite element method. Stress concentration zones have been identified.

The obtained results of testing the stress and strain state in structural elements with discontinuities of geometry can be recommended for use in practice. The conclusions obtained by examining this type of structure can be included in the process of designing new similar structures. The knowledge gained during the implementation of this paper can be directly applied to identify the behaviour of real structures in their working conditions.

REFERENCES

1. Pilkey, W.D., Pilkey, D.F., Peterson's Stress Concentration Factors, 3rd Ed., John Wiley & Sons, Inc., 2008.
2. Maneski, T., Milošević-Mitić, V., Ostrić D., Postavke čvrstoće konstrukcija (Preferences for the Strength of Structures, in Serbian), University of Belgrade, Faculty of Mech. Eng., Belgrade, 2002.
3. Đurđević, Đ., Anđelić, N., Maneski, T., et al. (2019), *Numerical-experimental determination of stress and deformation state in connecting lugs with the effect of contact area size*, Tech. Gazette, 26(2): 380-384. doi: 10.17559/TV-20180122105145
4. Ju, S.H., Rowlands, R.E. (1999), *A three-dimensional frictional contact element whose stiffness matrix is symmetric*, J Appl. Mech. 66(2): 460-467. doi: 10.1115/1.2791070
5. Antoni, N., Gaisne, F. (2011), *Analytical modelling for static stress analysis of pin-loaded lugs with bush fitting*, Appl. Math. Model. 35(1): 1-21. doi: 10.1016/j.apm.2010.05.002
6. Shi, G., Shi, Y., Wang, Y., Bradford, M.A. (2008), *Numerical simulation of steel pretensioned bolted end-plate connections of different types and details*, Eng. Struct. 30(10): 2677-2686. doi: 10.1016/j.engstruct.2008.02.013
7. Čelović, S., Tipsarević, M., Maneski, T., et al. (2015), *Numerical-experimental analysis of the foldable containers strength*, Tech. Gazette, 22(6): 1527-1532. doi: 10.17559/TV-20150112100813
8. Bajić, D., Momčilović, N., Maneski, T., et al. (2017), *Numerical and experimental determination of stress concentration factor for a pipe branch model*, Technical Gazette, 24(3): 687-692. doi: 10.17559/TV-20151126222916
9. Maneski, T., Computer Modelling and Design of Structures (in Serbian), University of Belgrade, Faculty of Mechanical Engineering, Belgrade, 1998.
10. Software KOMIPS®, University of Belgrade, Faculty of Mechanical Engineering, Belgrade.
11. Software ARAMIS®, GOM, Germany
12. Sedmak, A., Milošević, M., Mitrović, N., et al. (2012), *Digital image correlation in experimental mechanical analysis*, Struct. Integ. and Life, 12(1): 39-42.