23rd International Conference
ENGINEERING MECHANICS 2017
Svratka, Czech Republic, 15 – 18 May 2017

THIN WALLED PIPE CRUSHED IN JAWS

S. Sławski[*], S. Duda[**], T. Machoczek[***]

Abstract: *In the paper the authors present the process of creating physical model. The experiment consisted in crushing the sample in the jaws of a testing machine. The values of force were selected to enable verification of the physical model. During the experiment the strain in the areas under strain gauges and displacements of selected points were observed. The material properties of the tested element were determined by performing static tensile test.*

Keywords: Finite element method, True stress – strain curve, Strain gauge, Static tensile test.

1. Introduction

Conducting reliable numerical simulation requires a well-validated physical model. In the paper the authors present process of creating physical model. In order to validate it an experiment has been performed where displacement and strain were recorded. The authors compared results from the experiment and numerical simulation developed in LS Dyna.

2. Experiment

The experiment which relays on crushing a sample in testing machine MTS 858 Table Top System was performed. The dimensions of steel sample which have been used in experiment were 52 mm of diameter and 2.5 mm wall thickness. The sample was 50 mm long. On front surface of the sample markers were applied. Their positions and numeration were shown in Fig. 1. Those markers were used to determine displacement of these points based on video which was recorded during the experiment. On the inside and outside surface of sample three strain gauges were glued. Glued strain gauges were product of Vishay, type EA-13-120LZ-120/E. Numeration and position of those strain gauges were shown in Fig. 2.

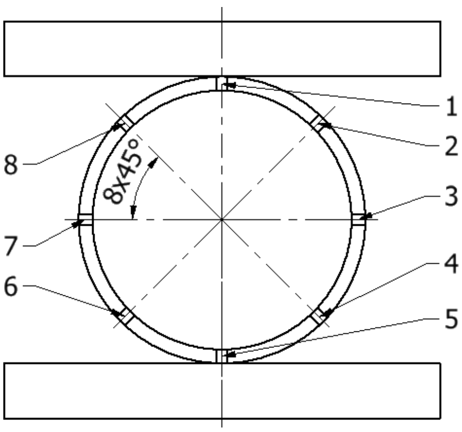

Fig. 1: Numeration and positions of markers.

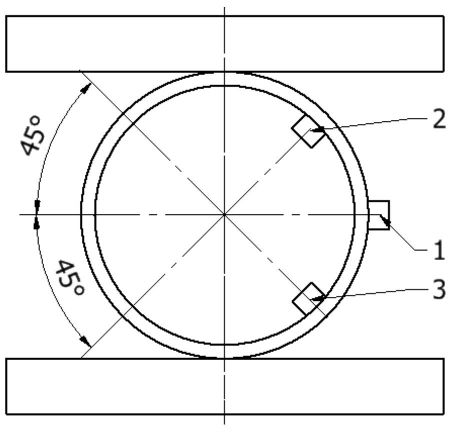

Fig. 2: Numeration and positons of strain gauges.

[*] M.Sc. Eng. Sebastian Sławski: Institute of Theoretical and Applied Mechanics, Silesian University of Technology, Konarskiego 18a; 44 100, Gliwice; PL, sebastian.slawski@polsl.pl

[**] Assoc. Prof. Sławomir Duda, PhD.: Institute of Theoretical and Applied Mechanics, Silesian University of Technology, Konarskiego 18a; 44 100, Gliwice; PL, slawomir.duda@polsl.pl

[***] PhD. Eng. Tomasz Machoczek: Institute of Theoretical and Applied Mechanics, Silesian University of Technology, Konarskiego 18a; 44 100, Gliwice; PL, tomasz.machoczek@polsl.pl

The strain values of strain gauges during the experiment were shown in Fig. 4.

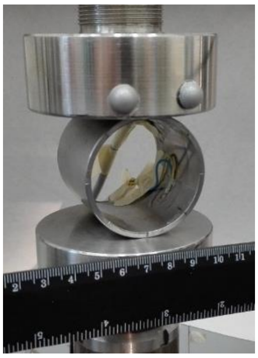

Fig. 3: Experiment – sample between testing machine jaws.

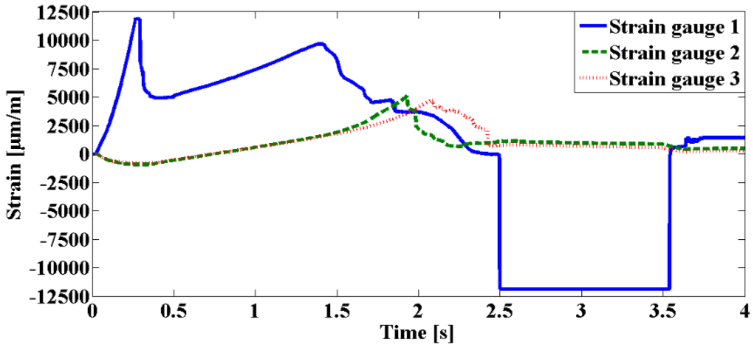

Fig. 4: Values of strain gauges strain.

In order to recognize parameters of sample material a static tensile test has been performed. Dimension of the sample were in accordance with the applicable standard. The static tensile test was performed on MTS 858 Table Top System testing machine. Basing on the sample dimensions and registered data from testing machine engineering stress – strain curve was determined (Jakubowicz et al., 1984). The next step was to transform engineering stress – strain curve to true stress – strain curve using equations 1 and 2.

$$\sigma_T = \sigma \cdot (1+e), \tag{1}$$

$$e_T = \ln(1+e), \tag{2}$$

where: σ – enginnering stress [MPa],

σ_T – true stress [MPa],

e – enginnering strain,

e_T – true strain.

The determined true stress – strain curve has been shown in Fig. 6.

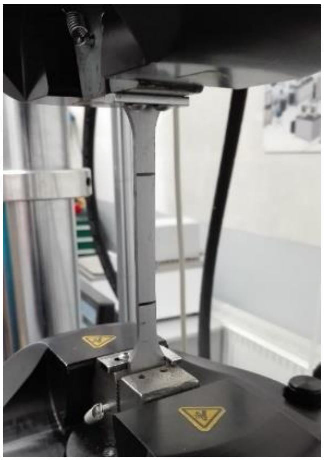

Fig. 5: Static tensile test.

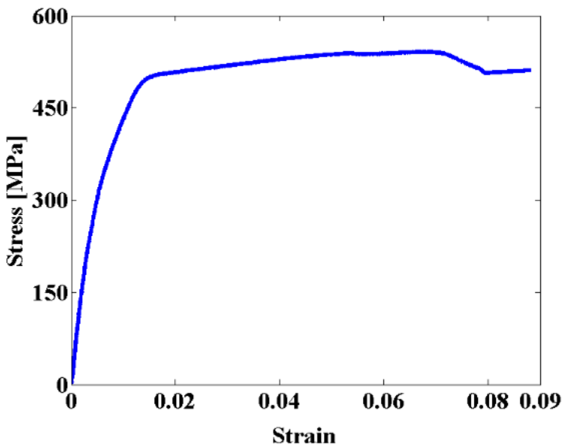

Fig. 6: True stress – strain curve.

Based on true stress – strain curve value of Young's ratio was determined using Eq. (3).

$$E = (\Delta F \cdot L_0) / (S_0 \cdot \Delta L), \tag{3}$$

where: ΔF – force increase [N],

S_0 – cross sectional area of sample [mm^2],

L_0 – measuring length [mm],

ΔL - measuring length increase.

Young's ratio value has been determined as 234 GPa.

3. Numerical simulation

A numerical model was created using LS PrePost. The model consists of three parts: thin walled pipe and two circular surfaces. Circular surfaces are simplification of testing machine jaws. Maximum finite element size for circular surfaces is 2 mm. Maximum finite elements size for pipe is 1 mm. During discretization process 11538 nodes and 11284 shell finite elements were created. Contact between created parts was defined as *AUTOMATIC_SURFACE_TO_SURFACE* with 0.15 static coefficient of friction and 0.1 dynamic coefficient of friction (Zienkiewicz, 1972). Jaws material was defined as *RIGID* with standard steel properties. Pipe material was defined as *PICEWISE_LINEAR_PLASTICITY*. Curve defining stress versus plastic strain was defined (Fig. 9). Boundary conditions which were applied received all degrees of freedom for one node of the motionless jaw and received degrees of freedom without movement in Z axis for one node of movable jaw. Load was defined based on data from testing machine which was recorded during the experiment. Curve defining force values were defined and shown in Fig. 8. Load was applied to movable jaw by *RIGID_BODY*.

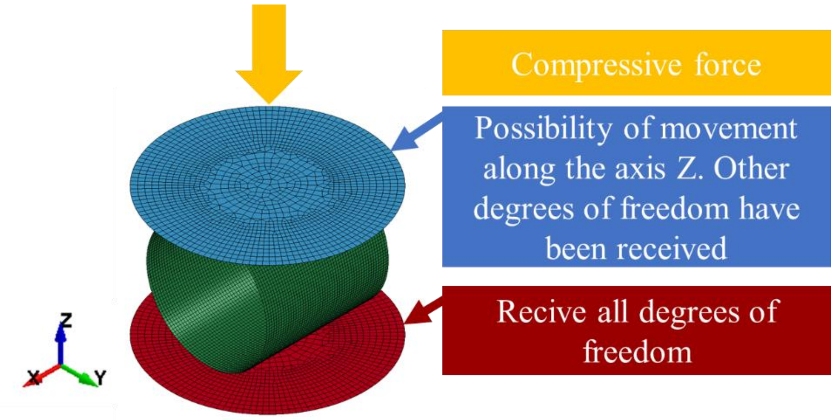

Fig. 7: Finite elements grid and boundary conditions.

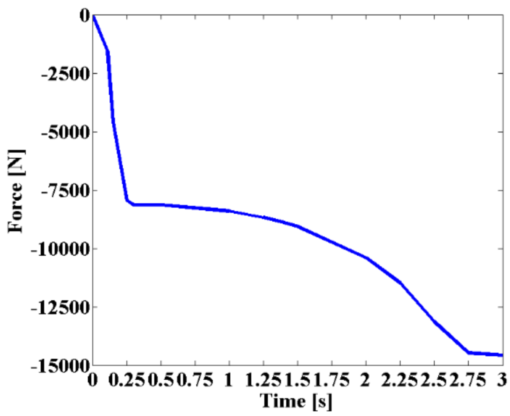

Fig. 8: Values of compressive force.

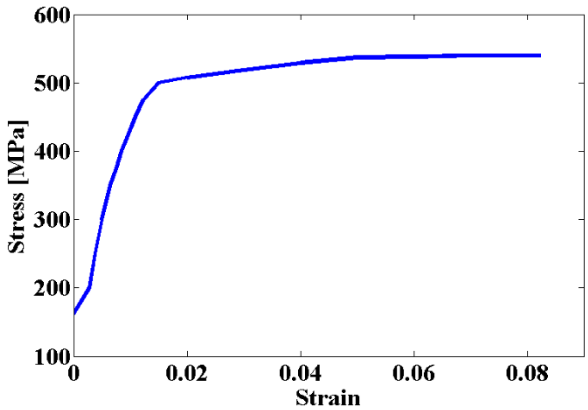

Fig. 9: True stress – plastic strain curve.

4. Results and conclusions

Displacement values for selected points have been compared. Fig. 10 shows compared values of displacement of selected points in a few moments of time while crushing the sample. Behavior of pipe from numerical simulation is a little bit different as compared to its real counterpart. Displacement of marker 1 in vertical axis is different. It results from type and size of finite elements which have been used. The size of shell finite element should be greater than its thickness. In this case the size of shell finite elements was lower than its thickness because the authors try to have most precise results which they could get using those types of finite element. The authors use shell finite elements also because of time which is needed to receive the solution. Receiving the solution using solid finite elements with the same size is few times longer. Average value of relative error calculated in relation to values of displacement from experiment is 72 % for horizontal axis and 11 % for vertical axis. In this case value of displacement error in vertical axis is more important because this is main direction of deformation. Such high error value of horizontal axis is caused due to finite element size. Only 12.5 % differences in horizontal axis between displacement value from experiment and numerical simulation are higher than finite element size. Most of differences are few times smaller than finite element size. 50 % of differences of displacement in vertical axis are smaller than finite element size. Biggest displacement errors were for nodes corresponding to markers 2 and 8 because of the wrong shape of deformation of the sample in numerical simulation. Strain and displacement between nodes are calculated using shape functions which based on result in nodes. That fact suggests that finite elements grid should be denser. Behavior of nodes corresponding to markers 1 and 5 showed restrictions of using shell finite elements and the reason why their size should be greater than theirs thickness. Deformation of the sample could not be as big as in the experiment because of the size of used finite elements. Summary of results from experiment, numerical simulation with shell type of finite element and numerical simulation with solid type of finite elements were shown in Fig. 11 (Sławski et al., 2017). In the case of solid finite elements deformation of the top of the sample looks like in the experiment.

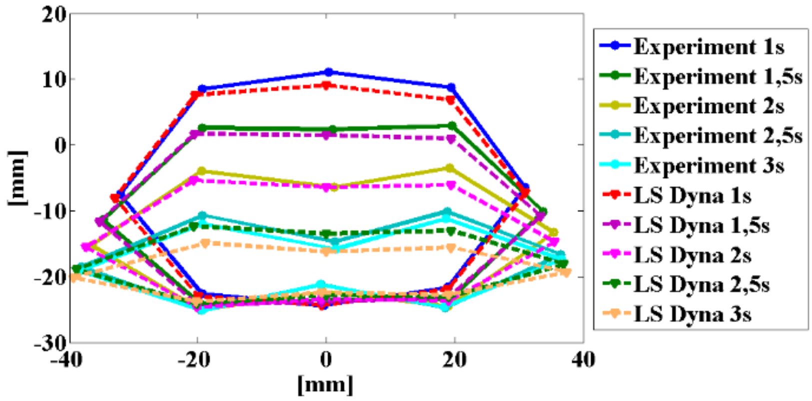

Fig. 10:Deformation of sample based on displacement of the markers in selected moment of time.

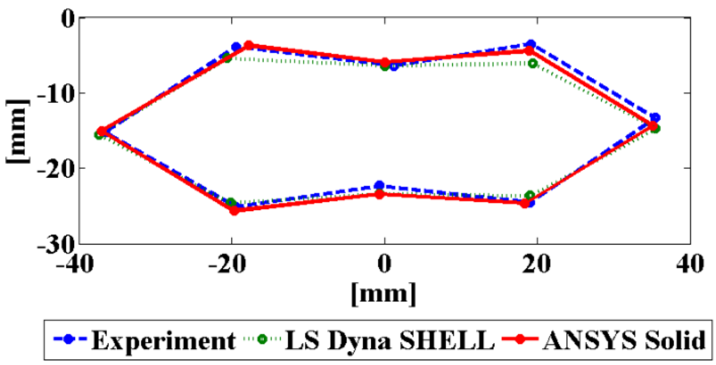

Fig. 11:Compare deformation of the sample in 2s time of the experiment.

The developed physical model could be considered as responding to its real counterpart. The model can be used to test behavior of pipe for different boundary conditions. Using shell finite elements in numerical simulations gives good results relative to time which is needed to solve a problem. Simulation engineer must be aware of restrictions which are associated with using this type of finite elements.

References

Sławski, S., Duda, S. and Machoczek, T. (2017) Determination of the state of effort in crushed thin walled pipes. Modelowanie Inżynierskie, Vol. 30, No. 61 (in Polish).

Jakubowicz, A. and Orłoś, Z. (1984) Strength of materials, Wydawnictwa Naukowo – Techniczne, Warszawa (in Polish).

Zienkiewicz, O. (1972) The finite element method, Wydawnictwo Arkady, Warszawa (in Polish).