FEM model and experimental measurement of clinched joint

P Malý, F Lopot and J Sojka

Czech Technical University in Prague, Faculty of Mechanical Engineering, Department of Designing and Machine Components, Technická 4, 166 07 Prague 6, Czech Republic

E-mail: pavel.maly@fs.cvut.cz

Abstract. This work describes creation of FE model of clinched joint and brings information about its verification with help of experiment. Also the sensitivity of the simulation model to the selected parameters (size of the model and coefficient of friction) and boundary conditions is studied so that the model is close to the reality. Deformation and uniaxial stress near the joint obtained with help of strain gauge is the output from the unidirectional, nondestructive test. Comparison of two values of stress in the chosen place obtained by experiment and FE simulation is the principle for verification of FE model and its adjustment. Based on the results it can be said that the FE model corresponds with the reality with the desired accuracy.

1 Introduction

Clinched joining of sheet-metal plates is a technology permanently applied in many branches of production thanks to many advantages that are e.g. high productivity, outstanding quality and its repeatability or possibility to connect sheet-metal plates with surface treatment. It is the reason why this type of joining material is studied in many articles where the process or mechanical properties are described. More information can be found e.g. in [1], simulation or experimental methods are mentioned in [2], [3], [4] and [5]. The other general example of application of numerical and experimental analysis is presented in [6]. This study concerns with the round clinched joints (figure 1) and describes the process of creation and study of FE model and its properties with usage of an experiment. This text brings information that is important for understanding of clinched joint properties from the simulation approach point of view.

Figure 1. Round clinched joint, [1].

2 Method

The work is solved in two related approaches: FE model of the clinched joint as a first approach is designed to correspond with the second approach, with the experiment.

2.1 Experiment
The set of 5 specimens was subjected to the experimental measurement to get the information about behavior of the joint under the real loading in the perpendicular direction to the axis of the joint.

2.1.1 Measurement protocol and instrumentation. Each specimen was subjected to the uniaxial, unidirectional, nondestructive test. The controlled variable was tension force acting on the specimen and smoothly increasing from 0 N up to 3000 N. The measurement was performed on the special single purpose testing stand that was set up for the experiment, the specimens were clamped in jaws, loading was applied with a hydraulic system and force was measured directly on the jaws with the force sensor HBM S9M/5kN.

2.1.2 Investigated set of specimens. The specimen consists of two sheet-metal plates from low carbon steel DX51D with cross-section 40×3 mm and length 280 mm connected in the middle by a round joint with characteristic diameter of 10 mm. The deformation near the joint on the upper sheet-metal plate in the direction of loading was measured using wire strain gauge from HBM – see figure 2.

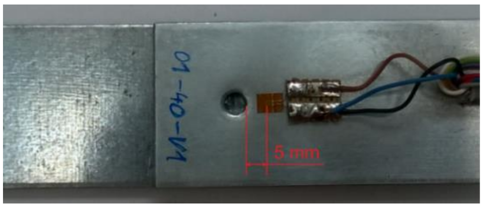

Figure 2. Strain gauge placement near the joint.

2.1.3 Data acquisition and processing. The test was controlled and the data were measured by National Instrument apparatus with LabView software. Data were processed by Matlab software.

2.1.4 Experiment outcomes. The resultant stress near the clinched joint in the place of the strain gauge for the loading force of 3000 N was obtained. It was compressive stress with the value of -29.5±2.6 N/mm^2 (average ± standard deviation).

2.2 Finite element modelling
Simulation model of the clinched joint is created as a deformable 3D model that consist of two plates connected by the joint. The clinched joint, i.e. interlock of upper and bottom sheet-metal plates, is created with help of curves with different radius. The shape is measured from the macrosection of the clinched joint with characteristic diameter of 10 mm (figure 3).

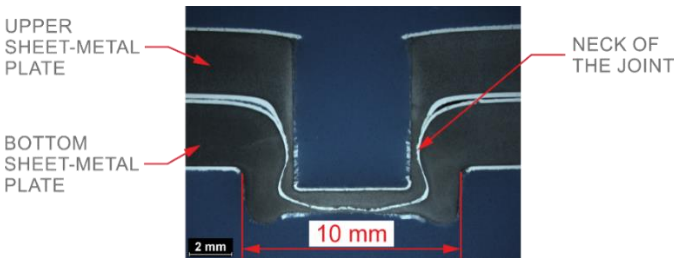

Figure 3. Macrosections of the clinched joints.

2.2.1 Model setup. The whole simulation model can be seen in figure 4. The symmetry of the model in the plane XY, i.e. the half of the model, is applied in the simulations. The dimensions of each plate are: length L = 75 mm or 280 mm, width of the half of the model W = 40 mm and the thickness T = 3 mm. The boundary conditions were applied in the reference points (RP-1 and RP-2) that are placed at the end of each plate in the middle of the thickness. The reference points are connected to the adjacent surfaces width length of 20 mm using continuum distributing couplings. The used material is steel with isotropic elastic behavior (Young's modulus 210 GPa, Poisson's ratio 0.3), the deformations are expected to be small, therefore nonlinear effects are not assumed in the calculation. Parts are meshed using 8-node linear bricks with reduced integration (C3D8R).

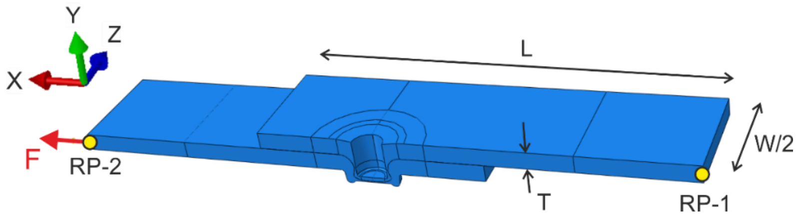

Figure 4. Whole model for simulation.

*2.2.2 Loading.*The applied force F is in X direction and has the value of 3000 N which corresponds to the load and conditions of experimental testing. The force is much lower than the limited force (shear strength) of the joint which is approximately 4000 N for the tested type and size of clinched joint, therefore no plastic deformation is assumed.

2.2.3 Joint model. The model of the clinched joint, i.e. the interlock between the sheet-metal plates, is created using tie connection in the bottom part of the joint. Standard hard surface-to-surface contact with tangential behavior (frictionless, or penalty friction with friction coefficient *f*) is used for the contact of other surfaces, see figure 5.

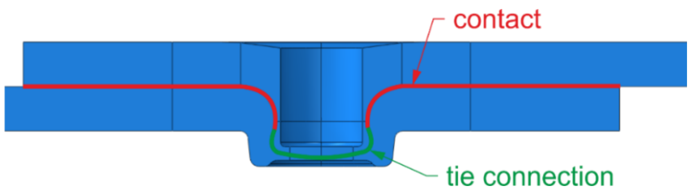

Figure 5. Model of the clinched joint.

2.2.4 Simulations. Set of ten simulations of the connection with different boundary conditions and parameters (length of the plate and friction) were done so that the model is analyzed and its robustness is verified. The simulations and the parameters are summarized in the table 1. The notation is used from Abaqus/CAE software.

Table 1. Simulation models and parameters

Simulations	Boundary condition RP-1	Boundary condition RP-2	Length L [mm]	Friction f [-]
C1	encastre	U2, U3, UR1, UR2, UR3	75	frictionless
C2	encastre	U2, U3, UR1, UR2, UR3	75	0.1
C3	encastre	U2, U3, UR1, UR2, UR3	75	0.2
C4	encastre	U2, U3, UR1, UR2	75	frictionless
C5	U1, U2, U3, UR1, UR2	U2, U3, UR1, UR2	75	frictionless
C6	encastre	U2, U3, UR1, UR2, UR3	280	frictionless
C7	encastre	U2, U3, UR1, UR2	280	frictionless
C8	U1, U2, U3, UR1, UR2	U2, U3, UR1, UR2	280	frictionless
C9	encastre	U2, U3, UR1, UR2, UR3	280	0.1
C10	encastre	U2, U3, UR1, UR2, UR3	280	0.2

Boundary condition in the reference point RP-1 is either encastre for the first case, or the displacement is restricted in all directions (X, Y, Z) and rotation is restricted only in the axes X and Y for the second case. The force F is applied in the reference point RP-2 hence the displacement in X direction is enabled for all simulations.
Other displacements and rotations in the point RP-2 are restricted for different simulation according to the table 1. Two sizes of the plates were used in the model, length L = 75 mm and L = 280 mm, the last one corresponds to the size of real testing specimens. The influence of the friction was verified by simulations for three cases, the frictionless contacts and penalty contact with constant friction coefficient of value 0.1 and 0.2.

3 Results

The resultant stress in X direction (S11) on the surface of the upper plate is chosen for analyzes and comparison of the joint model simulations. This direction corresponds to the direction of loading and the direction of the real stress measured by the strain gauge. The course of this stress S11 along the curve near the joint can be plotted and its value in the place of interest, i.e. the position of the strain gauge, is obtained. This value is used for evaluation and comparison of the simulations and verification with experimental measurement. The set of nodes and elements along the curve where the values of stress S11 are obtained can be seen in figure 6.

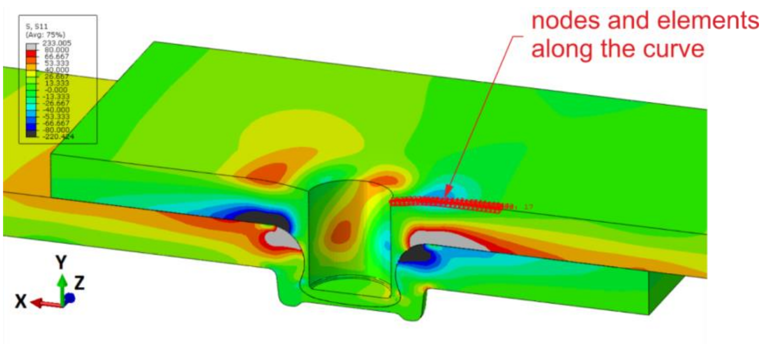

Figure 6. Nodes and elements along the curve for the stress S11

The example of the S11 development along the curve can be seen in figure 7. The position is measured from the edge of the hole of the joint on the upper plate as is shown for strain gauge position (figure 2).

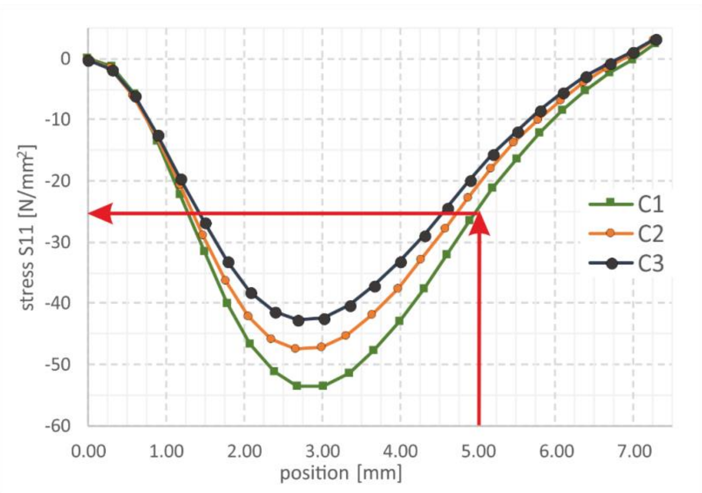

Figure 7. Development of S11 stress along the curve near the joint.

The calculated values of S11 for the all simulations in the place of interest (in the distance of 5 mm from the joint) are summarized in table 2.

Table 2. Result stress S11 [N/mm^2] in the position of the interest.

Simulation	C1	C2	C3	C4	C5	C6	C7	C8	C9	C10
S11 [N/mm^2]	-25	-21	-18	-22	-28	-32	-28	-34	-28	-24

The results of simulations using different plate size are summarized in table 3 where each pair of simulations (C1 vs. C6, C4 vs. C7 and C5 vs. C8) has the same boundary condition. It is obvious that the difference of the result stress is between 6 and 7 N/mm^2 which is approximately 20 %.

Table 3. Result stress S11 [N/mm^2] for different size of the plates.

	C1	C4	C5	C6	C7	C8	difference
Parameter: size, (C1 vs. C6)	-25			-32			7
Parameter: size, (C4 vs. C7)		-22			-28		6
Parameter: size, (C5 vs. C8)			-28			-34	6

Results of simulations with different friction are in table 4. All these simulations were performed with the same boundary conditions, the difference is assumed between adjacent values of friction (frictionless, $f = 0.1$ and $f = 0.2$) and has the value 3 and 4 N/mm^2 which is approximately 15 %.

Table 4. Result stress S11 [N/mm^2] for different friction.

	C1	C2	C3	C6	C9	C10	difference	
Parameter: friction, (C1 vs. C2 vs. C3)	-25	-21	-18				4	3
Parameter: friction, (C6 vs. C9 vs. C10)				-32	-28	-24	4	4

The influence of different boundary conditions for two lengths of the plates can be described by the results that are summarized in table 5. Comparison of results are focused on the size of plates with the length L = 280 mm because the simulation model is the same as the testing specimens. The difference between simulations C6 and C7 is 4 N/mm^2 (approx. 13 %), when the rotation in the place of acting force (RP-2) in axis Z is enabled for simulation C7. When the rotation in axis Z is enabled in the place of encastre (RP-1) in simulation C8, the difference between simulation C7 and C8 is 6 N/mm^2 (approx. 18 %). Thus, the change of boundary condition in RP-1 has greater influence than change of boundary condition in the place of acting force.

Table 5. Result stress S11 for different boundary conditions.

	C1	C4	C5	C6	C7	C8
Parameter: boundary conditions (C1vs. C4 vs. C5)	-25	-22	-28			
Parameter: boundary condition (C6 vs. C7 vs. C8)				-32	-28	-34

4 Conclusion

The FE model of clinched joint was created and analyzed using set of ten simulations. With respect to the application of the model, each simulation differs in chosen input parameters, i.e. size of the plate, friction and boundary conditions. Thus, the robustness and the sensitivity of the simulation model to the range of the input parameters were verified.

The change of the plate size influences the simulation quite significantly, so it is important to have the same dimensions of the testing specimen and simulation model. It can be said that the friction coefficient between surfaces in contacts is also important especially when the range of *f* is large. However, range of friction coefficient is often smaller than the one that was simulated. The results imply the necessity to choose the boundary conditions that correspond to the real conditions which can be often very difficult.

The verification of the FE model with the reality was done with help of experiment. The set of 5 specimens was subjected to the unidirectional, nondestructive test and the stress was obtained with help of strain gauge. Comparing of measured stress (-29.5±2.6 N/mm^2) and stress from simulations (-32 N/mm^2 for simulation C6 and -28 N/mm^2 for simulation C9) showed that the selected FE models correspond with the reality.

References

[1] TOX®-Product Range [online]. [cit. 2016-09-20]. Available on http://www.tox-de.com/assets/countries/EN/pdf/TOX_Product_Range_00_en.pdf
[2] Coppieters S 2012 Experimental and numerical study of clinched connections. PhD Thesis, Katholieke Universiteit Leuven – Faculty of Engineering (Leuven, Belgium)
[3] Hamel V, Roelandt J M, Gacel J N and Schmit F 2000 Finite element modeling of clinch forming with automatic remeshing *Computers and Structures* vol 77 issue 2 pp 185-200 http://dx.doi.org/10.1016/S0045-7949(99)00207-2
[4] Oudjene M and Ben-Ayed L 2008 On the parametrical study of clinch joining of metallic sheets using the Taguchi method *Engineering Structures* vol 30 issue 6 pp 1782–1788 http://dx.doi.org/10.1016/j.engstruct.2007.10.017
[5] Xiaocong H 2010 Recent development in finite element analysis of clinched joints *Int J Adv Manuf Technol* vol 48 issue 5 pp 607-612 http://dx.doi.org/10.1007/s00170-009-2306-2
[6] Semrád K, Draganová K 2016 Numerical-experimental analysis of the tensile properties of the composite wing hinge connection of the ultra-light sport aircraft Int. Conf. on Měření, diagnostika, spolehlivost palubních soustav letadel (Brno: Univerzita obrany) pp 101-8 ISBN 978-80-7231-377-8