Numeric Simulation of Part Assembly During Pressing Within Assembly Production Process

Pavel Srb(✉), Josef Vosahlo, and Michal Petrů

Institute for Nanomaterials, Advanced Technologies and Innovation, Technical University of Liberec, Studentska 2, Liberec 461 17, Czech Republic
pavel.srb@tul.cz

Abstract. The article deals with numerical analysis for a study of the stress distribution and deformation of the component assembly during pressing within the assembly manufacturing process. The assembly design consists of a steel casing and a steel bottom and top lid. The fusion of the casing to the lids is realized through the plastic deformation of the structural elements of the casing. The quality of the combination of the parts is influenced by, among other things, the manufacturing tolerances of the individual semi-finished products, the shape of the forming tool and the course of the forming force. A numerical simulation was created using an explicit algorithm, where the corresponding marginal and initial conditions and time dependency of the manufacturing process were defined. The different parts of the assembly were in delicate contact, so as to predict the force action in the different parts during the manufacturing process. The model created can be used to upgrade the geometry of individual parts of the report as well as the forming tool and contribute to the optimization of the production process.

Keywords: Joining · Forming · FEM simulation

1 Introduction

Joining parts is a key element in the design of structures, detachable or non-detachable joints can be used. For design an electrically powered vehicle, great emphasis is placed on the total weight of the final structure and also the energy consumption of the technologies used. Forming is an important technology in the industrial production process. It combines low energy consumption and material use with good efficiency. Plastic deformation can be used to join two or more parts without the need of additional material [1]. A well-known example of the effective use of this connection is the production of a camshaft. In general, the camshaft can be manufactured in several ways, usually by forging or machining. Meusburger describes a production process of a folded camshaft where the individual cams are placed on a hollow shaft, which is subsequently plastically deformed, thereby connecting the cams and the shaft [2]. This procedure brings many advantages such as easier machining, lower material consumption and the possibility of using different materials. Joining by forming can be advantageously used in the construction of metal polymer sandwich materials [3]. Where is described an innovative method

J. Homišin et al. (Eds.): ICMD 2020, AHE 17, pp. 227–233, 2023.
https://doi.org/10.2991/978-94-6463-182-1_24

of joining perpendicular sandwich composite panels using a new three-stage joining by the forming process. This process is capable of producing mechanically locked joints with larger and stiffer flat-shaped heads than those fabricated by alternative single- or two-stage solutions.

FEM simulation is a suitable tool for the analysis of the forming process and its subsequent optimization. Parameters that are very difficult to measure can be analyzed. Jiang [4] describes a sophisticated FEM analysis of the rolling process with the consideration of friction variation models. FEM can also be used to investigate surface roughening in the forming process [5].

The aim of the study was analyzing the stress distribution and deformation of the component assembly during pressing within the assembly manufacturing process. A numerical simulation was created using an explicit algorithm, where the corresponding marginal and initial conditions and time dependency of the manufacturing process were defined. The results of the analysis will be used to improve of the production process.

2 Materials and Methods

The analyzed part is the metal casing of the electronic component used in the vehicle. Figure 1 shows analyzed part. The casing consists of three parts connected by a forming process performed on an automatic line. During the process, edges are pressed to form the final joint. This analysis deals with the formation of a connection between housing and pole plate using a forming head. The overall dimensions of the part are approximately 30 mm in diameter and 50 mm high.

2.1 Numerical Simulation

The numerical model was created in Abaqus explicit software (Fig. 2). The first was creating of plastic material model corresponding to real material. Only a few basic material parameters were listed in material sheets. To create a plastic material model, it was necessary to obtain plastic parameters. Material data contain values of nominal

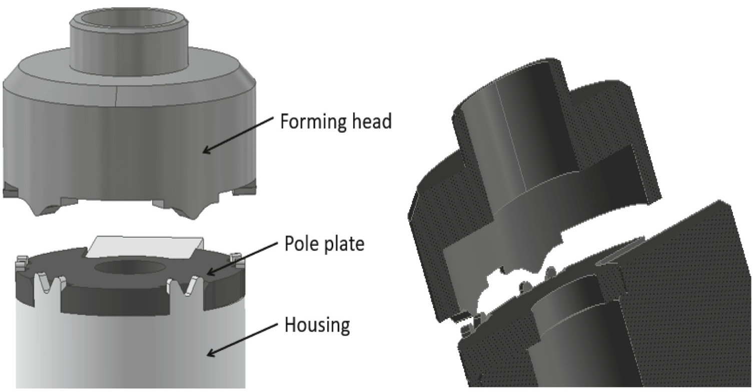

Fig. 1. Analyzed part.

stress and strain. Therefore, is necessary convert plastic material data from nominal stress-strain values to true stress strain-values [6]. The boundary conditions are shown in Fig. 3 the forming head is free in the vertical direction and the forming force acts there. The lower part of the part is fixed.

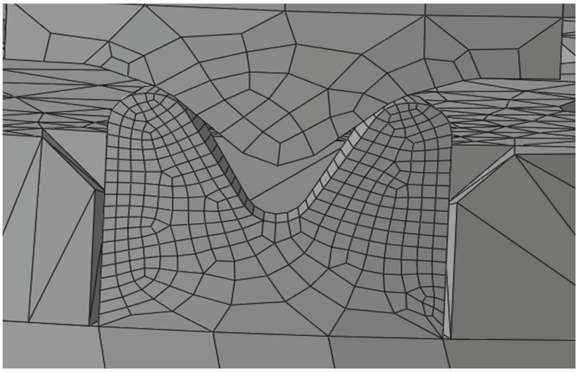

Fig. 2. Detail of FEM mesh.

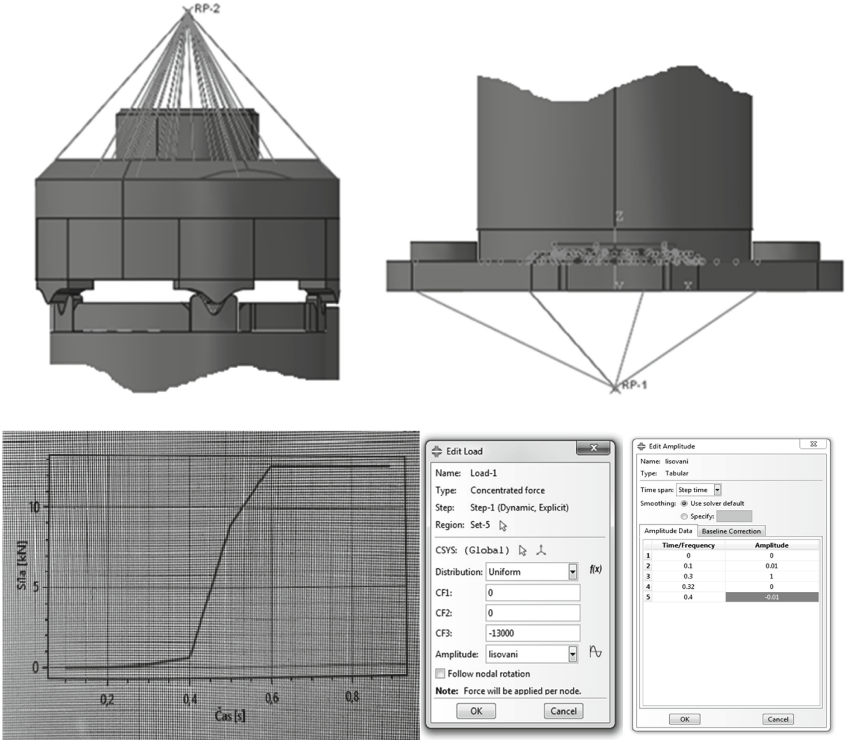

Fig. 3. Boundary conditions (above), forming force (below).

Table 1. Material parameters.

Material parameter	Value
Young's modulus	210 000 MPa
Poisson's ratio	0.3
Density	7 850 kg.m^{-3}
Yield stress	193 MPa
Ultimate strength	323 MPa
Ductility	58.2%

The relationship between true strain and nominal strain shows Eq. 1

$$\varepsilon_t = \ln(1 + \varepsilon_{nom}) \tag{1}$$

where ε_t is true strain and ε_{nom} is nominal strain. Equation 2 shows relationship between true stress and nominal stress.

$$\sigma_t = \sigma_{nom}(1 + \varepsilon_{nom}) \tag{2}$$

where σ_t is true stress and σ_{nom} is nominal stress. The metal plasticity model in Abaqus defines the post-yield behavior. The plastic data define the true yield stress of the material as a function of true plastic strain. The first piece of data given defines the initial yield stress of the material and, therefore, should have a plastic strain value of zero. Equation 3 shows relation of true plastic strain ε_{pl}.

$$\varepsilon_{pl} = \varepsilon_t - \frac{\sigma_t}{E}. \tag{3}$$

where E is Young's modulus. For material definition were available parameters shown in Table 1. Due to missing information of tensile test of material was for plasticity used only one next point except initial and it is $\sigma_t = 510$ MPa, $\varepsilon_{pl} = 0.456$.

3 Results and Discussion

A simulation of the molding of the pole plate of the part was performed according to the real production process, where the parts of the pole plate were pressed by the forming head with a force of 13 kN (Fig. 4). Performed numerical simulation and fitting of material models corresponds [7, 8] The following Fig. 5 shows a comparison of the part after forming the simulation and a real photo. The good agreement of simulation with the real part can be seen. In these figures can be seen good agreement of simulation with the real part. The Fig. 6 shows The distribution of deformations and stresses in the formed part.

The following figures show the distribution of deformations and stresses in the formed part (Fig. 6).

Fig. 4. Time response: Pressure by the forming head to housing.

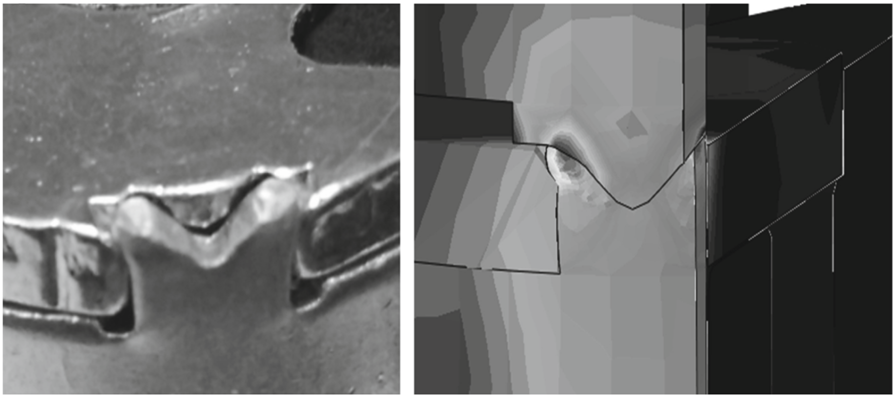

Fig. 5. Comparison of real part and numerical simulation.

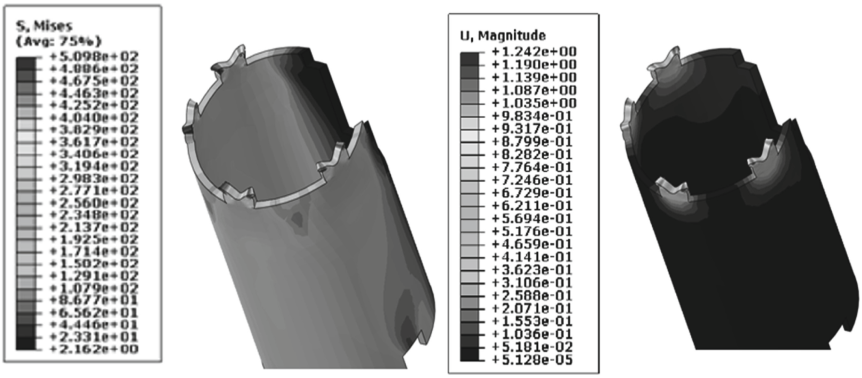

Fig. 6. The distribution of deformations and stresses in the formed part.

4 Conclusions

A simulation of the joining process by forming was performed. The simulation results show a very good agreement with the real production process. The results of the simulation can be used for future optimization of the production process and shape optimization of individual parts in order to achieve higher connection safety. Optimization can also lead to reduced weight and part cost.

Acknowledgements. The result was obtained through the financial support of the Ministry of Education, Youth and Sports of the Czech Republic and the European Union (European Structural and Investment Funds - Operational Programme Research, Development and Education) in the frames of the project “Modular platform for autonomous chassis of specialized electric vehicles for freight and equipment transportation”, Reg. No. CZ.02.1.01/0.0/0.0/16_025/0007293.

References

1. Groche, P., Wohletz, S., Brenneis, M.: Joining by forming—A review on joint mechanisms, applications and future trends, Journal of Materials Processing Technology, Volume 214, Issue 10, 2014
2. Meusburger, P.: Lightweight design in engine construction by use of assembled camshafts. MTZ Worldwide, Volume 67 (2006)
3. Pragana, JP., Contreiras, TR., Bragança IM.: Joining by forming of metal–polymer sandwich composite panels. Proceedings of the Institution of Mechanical Engineers, Part B: Journal of Engineering Manufacture. 233(10), (2019)
4. Jiang, Z., Xie, H.: Application of Finite Element Analysis in Multiscale Metal Forming Process, Finite Element Method (2018), https://doi.org/10.5772/intechopen.71880
5. Ma, X., Zhao, J., Du,W.: Analysis of surface roughness evolution of ferritic stainless steel using crystal plasticity finite element method, Journal of Materials Research and Technology, Volume 8, Issue 3 (2019)
6. Hibbit et al. ABAQUS/CAE 6.13 User’s Manual, Dassault Systemes, 2013
7. Cherouat A., Borouchaki, H., Zhang J. Simulation of Sheet Metal Forming Processes Using a Fully Rheological-Damage Constitutive Model Coupling and a Specific 3D Remeshing Method, Metals, volume 8, Issue 991; (2018), doi:https://doi.org/10.3390/met8120991
8. Yanamundra, K., Karthikeyan, R., Naranje, V.: Finite element simulation and Experimental verification of Incremental Sheet metal Forming, Materials Science and Engineering 346 (2018) 012075 doi:https://doi.org/10.1088/1757-899X/346/1/012075