Applied Computer Science, vol. 12, no. 1, pp. 40–47

Submitted: 2016-02-10
Revised: 2016-03-03
Accepted: 2016-03-12

Abaqus, Deform 3D, Numerical analysis, Stress

Patryk RÓŻYŁO[*], Łukasz WÓJCIK[**], Katarzyna WRZESIŃSKA[***]

NUMERICAL ANALYSIS OF AN ALUMINIUM PROFILE UNDER BENDING USING ABAQUS AND DEFORM-3D

Abstract

The paper describes the process of symmetric bending of an aluminium specimen using two independent environments for numerical analysis. A FEM analysis is performed on an aluminium channel section. A comparison is made between the bending process performed in the Abaqus environment and that run with the Deform-3D system. In both numerical analysis systems, the crucial point is to determine the state of reduced stresses for a designed computer model. The paper provides a visual comparison of the profiles subjected to bending; it also determines displacements and stresses in the tested structures. To perform a nonlinear static analysis, we define a material model, determine boundary conditions, interactions as well as generate a mesh. The results produced with the two numerical systems agree to a very high extent.

1. INTRODUCTION

Aluminium channel section profiles are widely used in a number of building fields and mechanical engineering. These profiles are frequently used to ensure support and higher rigidity of structures. An advantage of using channel section profiles is their relatively high weight strength compared to their relatively small weight. The most widely used profiles are made of aluminium. Irrespective of their length, channel section profiles can carry exceptionally heavy loads, that is why they are commonly applied. The situation changes when these structures are exposed to loads that cause bending. A great deal of studies mainly

[*] Lublin University of Technology, Nadbystrzycka 36, 20-618 Lublin, p.rozylo@pollub.pl
[**] Lublin University of Technology, Nadbystrzycka 36, 20-618 Lublin, l.wojcik@pollub.pl
[***] Maria Curie-Skłodowska University, Akademicka 19, 20-033 Lublin, katarzynawrzesinska@onet.eu

investigate the state of effort and instability for specified critical forces of channel section structures. Most analyses describe buckling states of thin-walled profiles subjected to precisely specified axial loads. In particular, the works [3, 4, 6, 7] investigate rigidity and buckling states for specifically defined boundary conditions. There is a scarce number of publications that deal directly with the problem of cross bending in open channel profiles. The studies [5, 8, 9] report the result of bending and stresses in channel section profiles obtained with Abaqus 6.14. Profile forming is very often used to produce decorative elements and frame constructions for the automotive industry. The process of forming aluminium profiles is described in the work [2]. Other publications report the numerical results of bending processes performed using Deform-3D [1, 10, 11]. The numerical environments described in this paper are two totally independent simulation packages. Abaqus is a system for numerical analysis of strength, thermal and electric processes, flow analysis etc. Deform-3D is mainly used for plastic analysis and formation of plasticized structures. The two programs cannot be operated jointly, nor do they enable rapid file sharing due to their totally different numerical environments. The objective of this paper was to compare the state of strength of two identical models of channel section profile. The analysis of a plastic-elastic model takes into account non-linear behaviour of material. Due to reaching the yield point, the material's rigidity changes in relation to the elastic range. The numerical simulations did not took account of material temperature or the temperature affecting the model from the outside. In terms of material strength, bending is a state of object deformation wherein a straight non-deformed beam becomes curved after the application of load. There are numerous theories about types of bending and relationships describing this process. Nonetheless, this paper focuses on investigating stresses and their distribution within the entire model. Although beam resistance to transverse loads has been studies for a very long time, solving it using two totally independent systems is an innovative approach to this problem.

2. MATERIALS AND METHODS

The investigation was performed on a channel section beam made of aluminium. The beam had a length of 250 mm, a thickness of 2 mm and the standard dimensions of 40x20 mm. The material model was ascribed the properties of PA45 aluminium, also denoted as 6061. Young’s modulus was 70000 MPa, the Poisson ratio was 0.33. The yield point R_e, was 250 MPa, the maximum strength R_m, was 300 MPa, while the elongation at maximum strength was roughly set to 9 %. The material properties of the described model were defined in an identical way for both numerical environments. Description of the analysis was consistent with the mathematical relationships presented below, where σ – stress, ε – strain, v – strain rate, T – temperature.

$$\sigma = f(\varepsilon, v, T) \tag{1}$$

Material properties defined in Abaqus are described above, while the plastic properties assigned in the Deform-3D system are listed in a figure below.

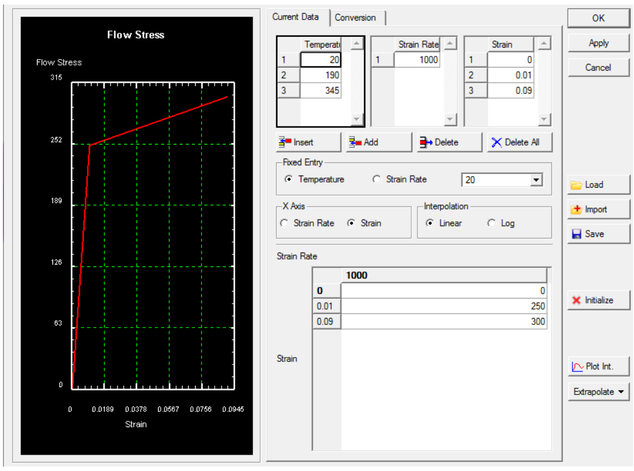

Fig. 1. Plastic characteristics of the tested material [source: own study]

A table given below lists the chemical composition of the modelled aluminium alloy.

Tab. 1. Chemical composition of aluminium alloy [1]

Chemical composition [%]							
Si	Fe	Cu	Mn	Mg	Cr	Zn	Ti
0.3- 0.6	0.1- 0.3	max 0.1	max 0.1	0.35- 0.6	max 0.05	max 0.15	max 0.1

The tested material was characterized by good strength, fatigue strength and bending strength. Aluminium PA45 is an Al – Mg – Si – Cu alloy. This alloy is subjected to supersaturation and artificial ageing at a suitable temperature and for a suitable time. Owing to its properties, this alloy can be easily subjected to metal forming operations such as bending and press forming, which enables forming complicated shapes of profiles and sheet metal plates.

To perform Deform-3D calculations, the material properties of aluminium are imported from external tables. To perform a numerical simulation in Deform-3D, it was necessary to create spatial models using some other CAD program, since Deform-3D only enables the design of basic geometric solids, such as cuboid and cylinder models. However, models of solids designed in other programs can be imported to Deform-3D into a CAD file. We designed the models of solids using Solid Edge ST7, and then exported them to files with the extension *.stl. Following the import of the models and fixing them in a way that reflected the real system, the channel section solid was assigned relevant plastic-elastic material properties via creation of a material model. Next, we indicated the regions of surface interaction and set a friction factor between the tools and aluminium (μ=0.2). The channel section profile was fixed in the measuring system such that the forming tool presses on the lower surface of the element. The profile was subjected to bending by a 20 mm diameter roll. The boundary conditions describing the motion of the tools were set such to reflect the real behaviour. The profile-bending roll moves at a constant velocity of 2 mm/s (which must be defined in the program) over a distance of 5 mm. The profile of a thin-walled channel section was mounted on two fully fixed supports.

The numerical simulation in the Abaqus environment was performed in a similar manner. First, we drew the channel section profile stretched to a desired length. Other subassemblies were also designed using this system. After that, we defined material properties of aluminium described earlier in this paper. Next, all the above were put together via functions available in the program in order to continue the FEM analysis. We determined the relationships pertaining to contact between cooperating tangential and normal surfaces for a friction factor typical of aluminium. The numerical model was examined only based on nonlinear static analysis in accordance with Newton's first law of motion where forces of inertia do not occur (i.e. the system does not accelerate). The boundary conditions were defined in the same way as in Deform-3D where the profile-supporting elements were fully fixed, while the 20 mm diameter roll underwent a uniform displacement in the opposite direction relative to the Y axis, which led to symmetric bending of the fully fixed thin-walled channel section profile.

The numerical models and their boundary conditions are shown in a figure below.

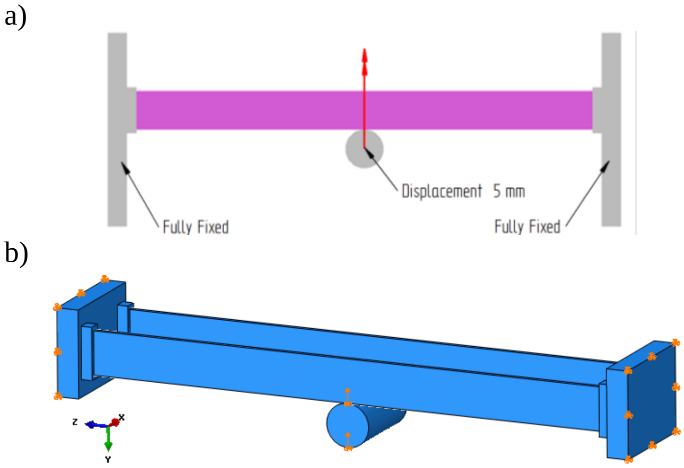

Fig. 2. Numerical model and its boundary conditions:
a) Deform-3D, b) Abaqus [source: own study]

The final stage of defining the numerical process involved generating a finite element mesh. The mesh in Deform-3D was defined according to the program's capacity (it was impossible to select any other mesh type). The mesh of the channel section was made up of 29671 tetragonal elements.

The discretization process was run differently in the Abaqus software. This program enables the user to determine a mesh depending on demand and results accuracy. Initially, the discretization involved model partitioning to obtain a structure with a uniform type of mesh element. Following the model partitioning, we obtained the best possible type of hexagonal elements – C3D8R (i.e. elements with three degrees of freedom, eight nodes and reduced integration). The mesh was made up of 15032 elements and 22992 nodes for the channel section, the supports and the bending roll. Despite the much smaller number of finite elements and thus a much lower density of this mesh, the results are similar to those of the tetragonal mesh obtained with Deform-3D. The FEM mesh generated in both environments for numerical analysis is shown in a figure below.

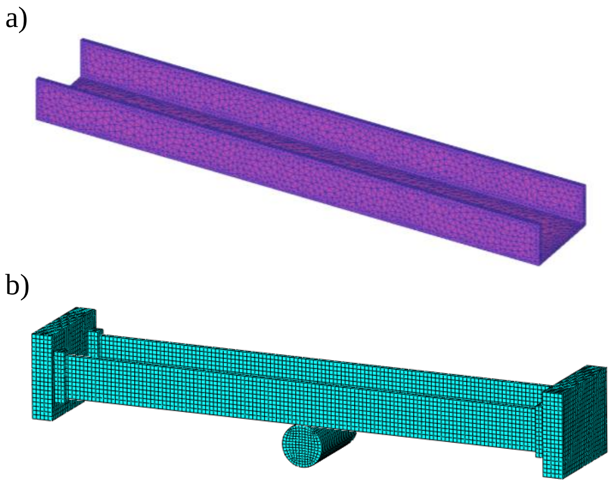

Fig. 3. Numerical model and it FEM mesh:
a) Deform-3D, b) Abaqus [source: own study]

3. RESULTS

This study investigated a bending process by the finite element method. In the analysis, we compared the tested material's strength properties and the numerical results obtained with two FEM-based simulation programs. The study involved investigation of reduced stresses and displacements at forced displacement of the profile-bending roll. Below we provide figures illustrating the agreement of the results produced with Deform-3D and Abaqus.

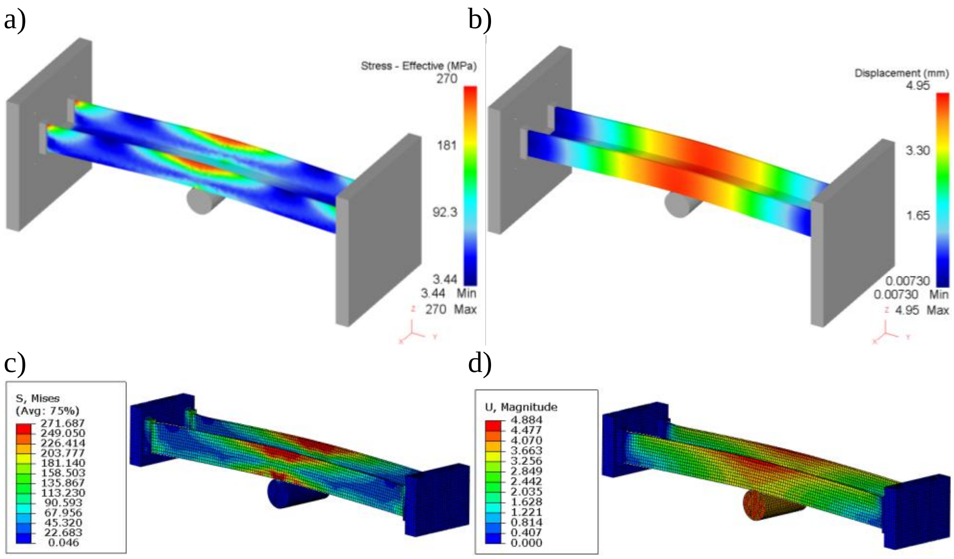

Fig. 4. Numerical results of stresses and displacements: a) effective stress in Deform-3D, b) displacement in Deform-3D, c) effective stress in Abaqus, d) displacement in Abaqus [source: own study]

The numerical results obtained with Abaqus and Deform-3D show very high agreement. The simulations performed in Deform-3D led to the production of a deformed profile made of aluminium alloy. The profile was subjected to bending to about 5 mm, which led to deformation of the profile's arms. The highest stresses can be observed in the upper part of the profile's arms and the effective stress is almost 270 MPa. The yield point is exceeded after the bending roll has travelled a distance of about 2.2 mm.

The Abaqus results show very high agreement. As the roll travels, the stresses gradually increase to reach 271.7 MPa at the maximum displacement of about 5 mm. At 2.23 mm, the stresses exceed the yield point.

The results of reduced stresses produced using two different numerical systems differ only by 0.6 %. By defining the problem via setting parameters of the model, material properties, interactions, boundary conditions and numerical meshing, we could compare different types of physical and mechanical problems. The high agreement between the FEM results means that the numerical computations are precise, irrespective of employed advanced systems for investigating different processes such as bending. Despite the fact the number of the FEM mesh elements (tetragonal elements) used in the Deform-3D model is almost twice as high, the results are similar to those of the hexagonal mesh model in Abaqus. The similarity of stresses and displacements indicates accuracy of the analysis results produced using two independent numerical environments.

4. CONCLUSIONS

Numerical analysis reflects real operating conditions very closely, hence it can provide a number of solutions to be implemented in practice. The research on thin-walled channel section structures under bending is widely used to determine a degree of profiles exertion with specified material properties. The simulations enabled determination of effective stress in crucial regions of the structure. The results can be visually presented to illustrate modes of the profile's deformation during the simulation.

The FEM analysis results have led to formulation of the following conclusions:

– the numerical results show very high correspondence despite the application of different computational procedures (nonlinear statics in Abaqus and kinetics in Deform-3D),
– the parallel results prove that the plastic-elastic properties of the material were defined in a similar way,
– the numerical results are similar despite the use of different types of FEM mesh,
– irrespectively of the applied computational procedure, the boundary conditions were defined in the same way for both programs (fixing, displacement),
– the Abaqus and Deform-3D results of reduced stress for the applied displacement of the bending roll are corresponding,
– the shape of beam deflection and the regions of concentration of the maximum stresses are similar.

The analysis of the bending process for a thin-walled aluminium channel section profile led to determination of stresses and displacements.

The applied simulation software packages based on different computational algorithms produced similar results. Despite the use of two independent programs, the obtained modes of beam deflection are similar and the results of the effective stress agree to a very high degree.

The results open up possibilities for analyzing similar problems as well as further investigation of the problem discussed in the paper, e.g. with respect to specimen failure.

The results are confronted with both the assigned material properties and the results obtained using two independent programmes for material strength testing, Abaqus and Deform-3D, which are ideal tools for investigating stresses and displacements in thin-walled open channel structures.

REFERENCES

[1] CHIBA R., YOSHIMURA M.: *Solid-state recycling of aluminium alloy swarf into c-channel by hot extrusion*, Journal of Manufacturing Processes, 17, 2015, p. 1–8.

[2] CHOI Y., YEO H.T., PARK J.H., OH G.H., PARK S.W.: *A study on press forming of automotive sub-frame parts using extruded aluminum profile*, Journal of Materials Processing Technology, 2007, p. 85–88.

[3] DEBSKI H., JONAK J.: *Failure analysis of thin-walled composite channel section columns*, Composite Structures, 132, 2015, p. 567–574.

[4] DEBSKI H., KUBIAK T., TETER A.: *Experimental investigation of channel-section composite profiles behavior with various sequences of plies subjected to static compression*, Thin-Walled Structures, 71, 2013, p. 147–154.

[5] KUBIAK T., KACZMAREK L.: *Estimation of load-carrying capacity for thin-walled composite beams*, Composite Structures, 119, 2015, p. 749–756.

[6] KUBIAK T., SAMBORSKI S., TETER A.: *Experimental investigation of failure process in compressed channel-section GFRP laminate columns assisted with the acoustic emission method*, Composite Structures, 133, 2015, p. 921–929.

[7] MANIA R. J., KOLAKOWSKI Z., BIENIAS J., JAKUBCZAK P., MAJERSKI K.: *Comparative study of FML profiles buckling and post buckling behavior under axial loading*, Composite Structures, 134, 2015, p. 216–225.

[8] PASZKIEWICZ M., KUBIAK T.: *Selected problems concerning determination of the buckling load of channel section beams and columns*, Thin-Walled Structures, 93, 2015, p. 112–121.

[9] RÓŻYŁO P.: Optimization of I-section profile design by the finite element method, Advances in Science and Technology Research Journal, 10(29), 2016, p.52–56.

[10] WANG D., XU K., JIANG Z., LI R., ZHANG X., ZHANG Y., QIN S.: *Research on the cracking of the thick steel plate in the bending process,* IEEE, 2014, p. 5020–5023.

[11] ZHANG L. H., ZHANG Z. C., LI S. Y., CUI H., CUI H. X.: *FE simulation and bending speed optimization of N-TR continuous grain flow forging process for solid heavy crankshaft,* Obróbka Plastyczna Metali, 2006, p. 3–13.

[12] Abaqus HTML Documentation.