Numerical modelling of the thinning behaviour of sheet metal parts

V Gál¹ and Zs Lukács²

¹ PhD Student, University of Miskolc, Hungary
² Associate Professor, University of Miskolc, Hungary

E-mail: metgv@uni-miskolc.hu

Abstract. In the automotive industry there are several software for numerical simulations to predict the behaviour of the material, which can reduce the expensive try-out experiments and dies cost. These software uses different mesh types. The goal of this work is to compare the different mesh element types in sheet metal forming simulations from the viewpoint of excess thinning. The benchmark of the 2018 NUMISHEET conference was the base of this comparison. This benchmark investigates the formability of a 2.8 mm thick, hot rolled sheet with 440 MPa tensile strength. The FE modelling of the process was made with thin shell-, thick shell-, and brick element types, thereafter; it was validated by the minimum thickness of the part based on physical data measurement. To better identify the differences due to the mesh elements, the thickness distribution of the part was investigated, too.

1. Introduction

Most parts in the automotive industry – particularly in the Body in White production – are produced from sheet metal. To achieve the RFT (Right for the First Time) production, the behavior of the material needs to be predicted. The more complex the geometry of the part is, the more complex stamping tool required. To increase the productivity and to reduce the expenses of tryout dies, the finite element simulation of the deformation of the sheet metal has a key role [1].

In sheet metal forming simulations commonly three set of parameters are used, to determine the plastic deformation. The Yield surface, which determines the onset of the plastic deformation. The Flow curves that show the material answer for the plastic deformation. Nowadays, in sheet metal forming, the Forming Limit Diagram (FLD) with the Forming Limit Curve (FLC) is regarded as the most important materials characteristics, which belongs to the limit strains resulting in the fracture of the material. The procedure to determine these constitutive parameters can be very difficult both with measurement, and theoretical methods.

The difficulty of determining these parameters are caused for example by the high strength materials (e.g. ultra high strength steel-s or high strength aluminium alloys) or the elevated temperatures measurement that required for hot forming simulations. All of these needs more difficult equipment. Today the problems with the measuring equipment for flow-curves (and of course the Yield-surfaces) considered as solved. However, the FLD’s depends much more things and has several drawbacks, like the strain path or sheet thickness dependence, or even the differences between the calculated and the measured limit strain thanks to the friction dependence [2].
The accurate description of these parameters is the main target of the numerical simulations, because only in this way the behavior of the material can be predicted. Despite the FLDs are the most widely used material parameter in automotive industry, in most cases the Flow curve and Yield-surface can be enough for FEM due to the general prescription for the maximum thinning 30%, which is determined in the automotive industry. Based on this, the cracking, or the excess thinning of a material is not acceptable.

To achieve a successful simulation, by the accurate material card (which include the Flow curve, the Yield-surface and the FLD), the real manufacturing environment (die geometries and kinematics) and the accurate boundary conditions need to be generated. Based on this, the importance of the geometry types of the applied mesh element unequivocal. Otherwise -as you can see in the Figure 1. and Figure 2.- there are some cases when the results of the simulation is far from the reality. (The initial sheet thickness of the Figures below was 2 mm.)

![Figure 1. Results of a complex drawing operation of a ‘Cone’ part with shell element](image1)

![Figure 2. The physical result of the operation](image2)

For sheet metal forming simulations, the thin shell is the commonly used element type. For simple deformation cases, this element is appropriate. Nevertheless, in the cases of complex forming, especially when there is a forming operation in the opposite of the mean direction, inaccuracy may occur. This is one of the reasons why the dedicated FEM codes give the opportunity to use different element types.

Since different types of mesh elements or meshing methods may cause different results [3][4], the main goal of this work to examine the effect of these for local thinning. The base of the comparison was the benchmark of the 2018 NUMISHEET conference. In this work, the thinning behavior of an industrial part, was investigated with different mesh element types. The thickness dependence on die height, and the distribution of a sheet thickness along the cross section was examined.

2. **Experimental setup**

To have a better knowledge of the huge difference between the simulation and the real part that has shown above, the critical phase of the deformation needed to be examined. The excess thinning and the greatest difference occurred at one embossment of the four. The first thing that can be seen from the Figure 1. and Figure 2., that to create these embossments opposite operation to the drawing direction needed. To examine the current operation cross-section of the dies and the sheet was investigated (Figure 3.). In accordance with the industrial experience, the excess thinning was caused, because in the second step of the forming, while the material has no possibility to flow-in because the die of the first drawing step holding it back.

The huge calculating time comes with the complex geometry justified the decision to find an easier geometry, which create the same deformation condition, but make the calculating time shorter, and in
the same time make the evaluation simpler too. The NUMISHEET conference, which is one of the largest conference in the field of numerical simulations of sheet metal forming, every second year has a benchmark project. To achieve this project the geometry of a part and tools, the appropriate constitutive equations, and other important simulation parameters used to be given.

One of the main tasks of the benchmark of 2018 NUMISHEET was the examination of an industrial part thinning behaviour in case of different die heights. Figure 4. shows the geometries of the benchmark tools, and it can be seen, that they can cause comparable deformation condition of the sheet, that has been shown in Figure 3.

![Figure 3. Cross-section of the critical part](image1)

![Figure 4. The setup of the NUMISHEET part](image2)

The blank had a diameter 246 mm and a thickness 2,8 mm, which is suitable for the measurement of thickness changing. The first step of the forming operation, the application of a 50 kN holding force with the stripper. After that, the top die starts moving downward with constant velocity of 10 mm/s. The deformation caused by the die can be separated into two stages. In the first stage, the outer flange of the part has been made, thereafter in the second stage, when no material flow occurs thanks to the cup wall and the blank holder force, the centre boss has been formed. Thanks to the first stage and the stripper, the thinning will be advanced, and a fractur will occur by the centre box.

The material data was given by the description of the benchmark. For the flow curve uniaxial tensile tests were performed. For the contours of plastic work biaxial tensile test with cruciform specimen in seven different linear stress paths were performed. Also, the results were given of the physical measurement the drawing operation show the context between the die height (H) and the minimum thickness of the sheet in the second stage when the drawing operation occurs till cracking.

3. Finite element simulation

The numerical simulations of the drawing process were carried to predict the necking behaviour using to different FE codes. First, the AutoForm FE code was used, because nowadays this program package is the most widely applied one for automotive sheet metal parts [1]. Secondly, the DEFORM, which is a process simulation system for forming simulation with solid elements.

The need of the two software can be explained with the mesh element type we planned to compare. For triangular thin shell element AutoForm R8 were used. For thick shell element also the AutoForm R8 was used since the software provides the opportunity to choose the element type before the simulation. The meshing was made automatically by the AutoForm, which use adaptive remeshing during the deformation with both element type in case of inadequate element size. The DEFORM FE code usually use tetrahedral element, but provide the brick element as an option too. In this case the brick element is better than the tetrahedral because of the number of elements in the thickness direction.

Three different element types in two different software were used. The number of element layers was 11 with both shell type and 9 for the brick mesh (because of the rule of the software). The generating of the simulation was almost the same, in the DEFORM and AutoForm^R8. The tool surfaces were imported from the benchmark and because its deformation will not playing a role, rigid type of material
was choose for those. Constant static friction coefficient of 0.15 was used between the blank and the dies. The same material properties (flow curve, and yield surface), and the same boundary condition were used for both software.

The main goal of the measurements was to compare the increase of the excess thinning both in virtual and the physical measured ways. Due to this, the occurrence of the cracking was eliminated, and the last step (the die height) of the examination was the same as the physical measurement in the benchmark.

4. Results and discussion

The goal of the forming simulations was to examine the thinning behaviour of a hot rolled sheet metal in case of complex drawing operation. The simulation results for the minimum sheet thickness with different die heights (‘H’ in the Figure 4.) can be seen in the Figure 5.

![Figure 5. The minimum thickness as a function of die height](image)

The decrease of the thickness at the beginning of the deformation is almost the same, but with the increasing deformation is different for each element type. It can be observed that with the developed deformation, the thin shell element show increasing difference from the physical measured points. At the end of the measurement (before the fracture occurs) the difference of thin shell from the physical measurement is more than 0.4 mm, so the maximum thinning is only 40% instead of almost 60%.

The thick shell compared to the thin has thickness values much closer to the measured ones. In the uniform strain path of the deformation, the thickness of the specimen change as in the physical measurement. It has also been observed, that the difference starts increase when the necking occurs. The difference between the physical and FE measurement was only 11% of the original sheet thickness at the end of the simulation.

The best results of all was given by the brick type element. The thickness values are moving with the physical measurement. Based on this it can be observed, that the simulation of the excess thinning shows a strong dependence of the used element type in case when the deformation is reserve direction to the main direction.

To have a better knowledge of the differences examination of the cross-section in the last measured point before the cracking occur has been made in case of the different mesh elements. Since there was no exact information of the thickness distribution of the current physical part, the best fit element type from the Figure 5. has been choose as a base for comparison. We have investigated the sheet thickness as the function of arc length measured from the axis of the symmetry.
The thickness comparison can be seen on the Figure 6, and it shows huge differences between the thickness distributions. It is important to notice, that the highest changes in the thickness occurs at the simulations with brick mesh in the A and the B phases.

![Figure 6. Thickness distribution as a function of the arc length](image)

The thick shell element approaches the distribution of the thickness, at the critical points, the local minimum can be seen as by the brick element, but the values of the thinning are smaller than it need to be. As it have been expected on the basis of measurement above, the thin shell element has the worst effect for the thinning in this case. The thickness distribution actually has no critical minimum, so the excess thinning has not developed at the illustrated A and B points. The almost permanent minimum thickness suggest that, in case of complex forming operation the mesh from thin shell makes a homogeneous distribution of the elongations of the elements. The phenomenon can be explained by the fact that the thin shell mesh does not consider the stresses in the thickness direction.

Taking the elements suitability into account in case of automotive application, the comparison of the calculating times also needed. The simulation with the thin shell need 8 minute, the thick shell 12 minute, the brick mesh 4 hour computing time, on the same computer setup.

5. Conclusion
In the present research work FE simulations of a benchmark of the 2018 NUMISHEET conference have been performed to study the effect of the different element types for the excess thinning. It has been observed in the FE simulations, that the thick shell element in the viewpoint of the change of the minimum thickness with the die height, and also presumably the thickness change along the cross-section, show better results, than the thin shell element. The inadequate efficiency of the thin shell element can be explained with that these element does not count the stress in the direction of the thickness. High reliability was achieved in the simulation of complex thinning with brick elements.

Based on the results it can be declared, that the applied elements have significant effect on the modelling of thinning behaviour depending on the complexities of the forming operations in case when there is a forming part of the operation which direction is reverse to the main direction. The best results
for complex forming operation with the brick element, the least acceptable result with shell element can be achieved in case of complex deformation.

To convince the above described results, simulation was carried out with the part geometry that can be seen in the Figure 1. and Figure 2. The results were given by thin shell elements were only compared with thick shell elements in the software AutoForm®R8. There was no possibility to check the results with brick mesh element, due to the system requirements and calculating time caused by the dimensions of the part. It can be clearly seen in the Figure 7., that the simulations with thick shell elements has been approached better the reality than the shell element. The difference between the real part and the results of the thick shell also can be noted, but the thick shell can give better warning (the excess thinning was more visible than for thin shell).

Figure 7. The difference of the thickness results between thin and thick elements with a real part

The simulations with thin shell elements has the shortest calculating time, so it is perfectly suitable in case of the generally used stretching kind operations. In case of complex operation, the brick element can provide the best fitting with the reality, but in the viewpoint of the calculating time of the simulation too, the best option for the automotive industry can be the thick shell element.

Acknowledgement
The described article/presentation/study was carried out as part of the EFOP-3.6.1-16-2016-00011 “Younger and Renewing University – Innovative Knowledge City – institutional development of the University of Miskolc aiming at intelligent specialisation” project implemented in the framework of the Szechenyi 2020 program. The realization of this project is supported by the European Union, co-financed by the European Social Fund.

References
[1] Tisza, M. (2013); Recent development trends in sheet metal forming Int. J. Microstructure and Materials Properties, Vol. 8, Nos. 1/2, 2013
[2] Béres. G, Weltsch. Z, Lukács. Zs, Tisza. M; Prediction of stress- and strain-based forming limits of automotive thin sheets by numerical, theoretical and experimental methods AIP Conference Proceedings 1960, 160002 (2018); doi: 10.1063/1.5035028
[3] Li Kaiping, A. M. Habraken, H. Bruneel (1995); Simulation of square-cup deep-drawing with different finite elements; Journal of Materials Processing Technology 50 (1995) 81-91
[4] Alessandro R, Roberto D’A, Saverio A; Comparison of Meshing Strategies in THR Finite Element Modelling. Materials 2019, 12, 2332; doi:10.3390/ma12132332