[1.]Dávid GÖNCZI

FINITE ELEMENT INVESTIGATION IN THE FORMING PROCESS OF ALUMINIUM AEROSOL CANS

[1.]Institute of Applied Mechanics, University of Miskolc, Miskolc, HUNGARY

Abstract: This paper deals with the numerical analysis of the forming process of thin-walled aluminium cans. Our aim is to investigate different finite element modelling techniques to tackle this highly nonlinear problem and to determine the reaction forces during the necking process of aerosol cans. The commercial finite element software Abaqus is used to carry out the calculations. The effect of several parameters on the reaction forces is investigated.
Keywords: FEM, aerosol cans, reaction forces, shell

INTRODUCTION

In recent years the demand of aluminum aerosol cans with complex shapes was continuously growing. Due to the importance of these package products it is recommended to use numerical simulations to analyse the forming process of the cans during the design phase. This way the manufacturers can be competitive on the market and produce more complex shapes.

We can find several books, such as [1], [2] or [3] on the mechanics and finite element modelling of thin shell structures. Papers [4] and [5] investigates the extrusion and forming processes, Belblidia et. al. [6], [7] presented finite element techniques to predict the stress state and burst pressure of aerosol cans. Paper [8] investigated various parameters of the shaping process using experimental data. Takeutshi [9] presented a few basic problems in the forming process of aluminum cans. Paper [10] dealt with the hydroforming of cans. Several works [11-14] tackle the buckling problems of thin shells, which is important to determine the limitations of shapes (geometry) of aerosol cans (that can be manufactured within certain waste product ratio).

AIMS AND DATA

One of the main problems of this process is that it is a highly non-linear mechanical problem, in which the emphasis is on thin-walled shells made from strain-hardened aluminum. A linearly elastic, linearly hardening constitutive law will be used for our models. This problem involves geometric and material non-linearities combined with contact.

During the shaping process of cans, the desired final geometry is always reached within several steps because if too much plastic deformation occurs in one step, buckling (crush/pan) happens resulting in waste product. If the number of forming steps could be minimized, it would mean more efficient manufacturing. Furthermore, there is a continuous demand to use the reserves in the material more and more efficiently, thus the reaction forces (and its ratio to the force that causes loss of stability) within a step is an important data. We can derive safety factors which reflect the reserve of the certain shaping phases.

In this paper, our aim is to calculate the reaction forces during the forming steps with different finite element modelling techniques. The advantages and disadvantages of the different models will be outlined via two forming steps. Furthermore, the effects of various parameters (such as the thickness of the shell, friction coefficient, shape of the forming tool) on the reaction forces will be investigated. The numerical calculations will be carried out by Abaqus 6.13.

The material of the investigated cans is aluminum Al99.5 (EN AW 1050), the Young modulus is E=73 GPa, the Poisson's ratio is 0.4, the yield stress is 120 MPa. The average friction coefficient during the forming process is 0.05 between the aluminum can and steel tool.

In our first numerical example a simple cylindrical can is investigated. The initial geometry can be seen in Figure 1. The thickness of the shell is 0.38 mm, the diameter is 45 mm, the diameter reduction is 2.6 mm in the forming step, and the length of the can is 200 mm. The piece is placed in a sleeve, it is constrained at its bottom flat surface against vertical displacement, at its side of 50 mm height, against radial motion.

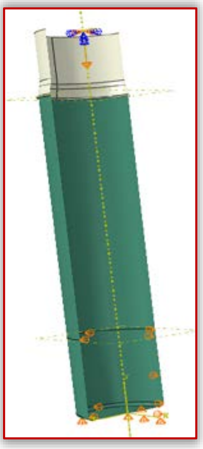

Figure 1. The sketch of the first example

In our second example, a necking step is investigated, where the change in diameter is 2.1 mm (from the initial 28.6 mm to 26.5 mm). The initial thickness of the shell is 0.48 mm, the initial geometry is more complex, and the boundary conditions are identical to the first example (Figure 2).

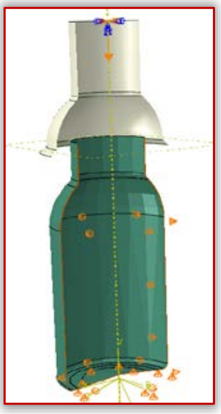

Figure 2. The sketch of the second example

There is a prescribed vertical displacement at the upmost edge of the tool.

MODELLING TECHNIQUES

Three models will be considered with displacement control. Alternatively we can use Risk method, especially if our aim is to include buckling. Due to the complexity of the contact between the can and the tool, it is necessary to pay attention to the contact. This means, that we need to initialize, control and stabilize the contact calculations and avoid errors, furthermore linear elements will be used.

The first technique uses three-dimensional geometry with shell elements. The tool is modelled as a discrete rigid part, the general contact algorithm of Abaqus is used. The advantage of this method over the other two models is that it can be used to investigate the loss of stability for the can.

In our second model, two-dimensional axisymmetric formulation is used, the can is modelled with shell element, the tools are represented by analytical rigid surfaces. We can use either the surface-to-surface or the general contact algorithms of Abaqus. In this case, the number of elements is minimal, the calculation is the fastest (compared to the other two methods).

The third method uses two-dimensional axisymmetric formulation, but the can is modelled with solid continuum elements. The advantage of this technique is the more accurate representation of the geometry, but the number of elements is much greater than in the second model due to locking considerations.

NUMERICAL EXAMPLES

In the first numerical example the diameter of a cylindrical can is reduced by 2.6 mm. According to experiments, the maximum reaction force during the shaping process is 650 N.

The model of the first method can be seen in Figure 3. After the convergence tests, the results are converged to 643N. The force-displacement diagram can be seen in Figure 6.

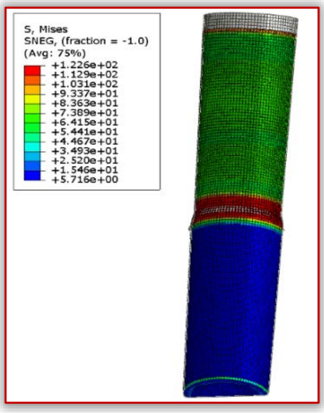

Figure 3. The deformed finite element mesh of the first example (with the von Mises stress distribution)

In our second method axisymmetric formulation is used, the can is modeled using shell elements. Figure 4 shows the model and the contact pressure (approximately 6 MPa) between the analytical rigid tool and the deformable can. In this case the maximum reaction force calculated at the reference point of the tool is approximately equal to the result coming from the previous model (Figure 6).

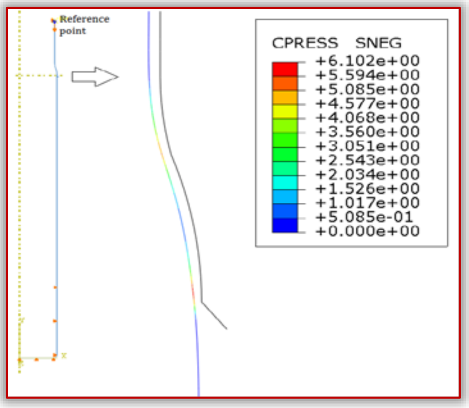

Figure 4. The assembly of the second method and the contact pressure during the forming process

The third modelling technique uses two-dimensional axisymmetric formulation with solid elements. Due to locking, small element size is used (Figure 5), the maximum reaction force is 668 N.

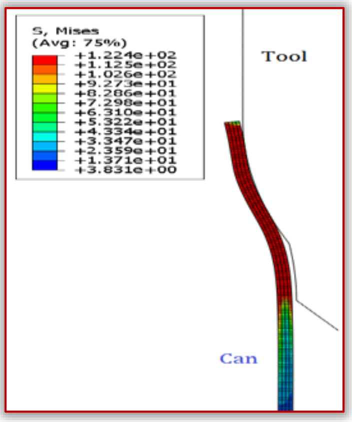

Figure 5. The deformed mesh of the first example using rigid tool and deformable can

Figure 6. The reaction forces during the forming process (green dots: 3D shell, blue line: axisymmetric solid, purple dots: axisymmetric shell)

Next let's considered deformable tool (instead of rigid surfaces), which results in slightly higher contact pressure (Figure 7), although the reaction forces are approximately the same. Furthermore, we modified the boundary conditions of the can, fixed the lower part of the cylindrical shell, the reaction forces did not change significantly.

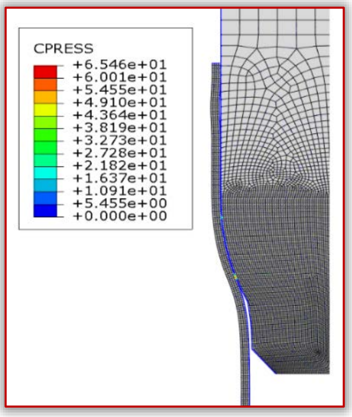

Figure 7. The contact pressure with deformable tool

In the second numerical example a necking process is investigated using three modelling approaches. The initial diameter of the can is 28.6 mm, the diameter reduction is 2.1 mm, the radius combination of the neck is R3.2 mm and R26 mm. According to the experiments, the maximum reaction force is 1.21 kN. Figure 8 presents the stresses coming from the three-dimensional shell formulation. The maximum reaction force is 1201N.

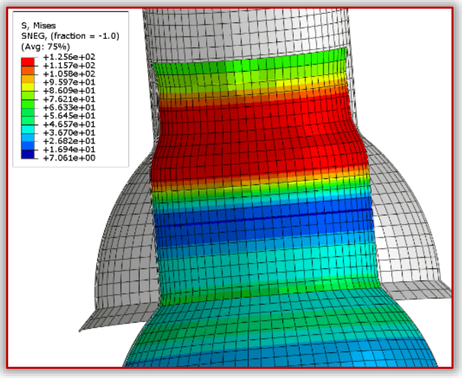

Figure 8. The von Mises stresses calculated from the second problem using the first method

The axisymmetric models can be seen in Figures 9 and 10. The reaction forces coming from the shell elements and continuum elements are equal (1195N). During the calculations of solid finite elements, the contact formulations of Abaqus failed when the sharp corner of the aluminum can touched the internal surface of the tool (even with contact stabilizations), that is why we had to modify the can geometry using a small fillet.

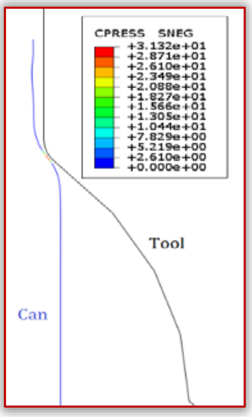

Figure 9. The deformed mesh of the second problem with axisymmetric shell element

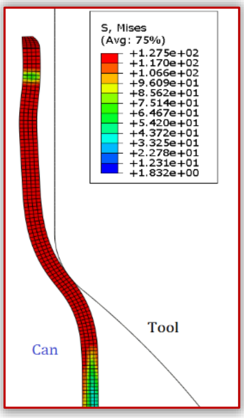

Figure 10. The deformed mesh of the second problem modelled with axisymmetric solid element and rigid tool

In the next step the friction coefficient, the shell thickness and the radius combination of the tool geometry are modified. Table 1 shows the results of the parametric investigation on the maximum reaction forces. When the friction coefficient or the shell thickness of the can increase, the forming forces increase too. With the increase of the initial radius of the tool, the maximum reaction force decreases, although the "spring back" effect is more significant.

Table 1. The effect of the friction coefficient, shell thickness and tool geometry on the maximum reaction forces

Friction coeffic.	Force (N)	Shell thickness	Force (N)	Radius (mm)	Force (N)
0.05	1200	0.48	1200	2.5	1200
0.15	1591	0.3	650	5	1040
0.3	2188	0.7	2230	15	890

Next, the crushing forces must be determined, where the loss of stability occurs. We can use either Riks-method or displacement controlled static analysis to determine the reaction force-displacement curve (and the crushing force form it). Obviously, we cannot use two-dimensional, axisymmetric formulation to tackle buckling problems, thus we have to use the first method (3D shell elements). Then we have to compare the crushing force and the maximum reaction force of the shaping process to evaluate the safety of the forming step. For example in our current case, a factor of safety can be defined as

$$n_c = \frac{F_{max}}{F_{crush}}, \tag{1}$$

which must be less than 1 and it reflect the „reserve" of the forming step. In our current example, the crushing force is 2.6 kN, which means that the factor of safety is 0.46. When the friction coefficient is greater, than 0.3 (dry friction between the can and tool) the factor of safety is more than 0.85, thus the number of waste increases (e.g. due to material or geometric imperfections).

CONCLUSIONS

The finite element analysis of the forming process for aluminum aerosol cans was investigated. Different methods were presented to calculate the maximum reaction forces, the force-displacement curves and to determine the feasibility of the forming step. The thin-walled cans were modeled with 3D shell elements, with 2D axisymmetric shell elements and finally with 2D axisymmetric solid elements. The results were compared to each other and to experimental results.

Acknowledgements

The described article 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] Reddy J.N.: Theory and Analysis of Elastic Plates and Shells. CRC Press, 2006.
[2] Chapelle D., Bathe C.J.: The Finite Element Analysis of Shells - Fundamentals. 2nd edition, Springer, 2011.
[3] Radwanska M., Stankiewicz A., Wosatko A., Pamin J..: Plate and Shell Structures: Selected Analytical and Finite Element Solutions. John Wiley and Sons Ltd., 2016.
[4] Patten S.: Design and optimisation of aluminium aerosol cans produced by the back extrusion process. MPhil thesis, University of Wales, Swansea, 2001.
[5] Hardy S.J., Abdusslam R.M.: Finite element modelling of the extrusion process for aluminium aerosol cans. Proc. IMechE, Part L, J. Materials: Design and Applications, 221, pp. 265-274, 2007.
[6] Belblidia, F., Corft, N., Hardy, S. J., Shakespeare, V., Chambers, R.: Simulation based aerosol can design under pressure and buckling loads and comparison with experimental trials, Materials and Design, 52, pp. 214-224, 2013.
[7] Belblidia, F., Corft, N., Hardy, S.J. , Bould, D.C. , Sienz, J.: Aerosol cans under pressure and buckling loads, Sustainable Design and Manufacturing, 1, pp. 13-17, 2014.
[8] Folle, L.F., Netto, S.E.S., Schaeffer, L.: Analysis of the manufacturing process of beverage cans using aluminum alloy, Journal of Material Processing Technology, 205, pp. 347-352, 2008.
[9] Takeutshi, H.: Numerical simulation technology for lightweight aluminium can. Journal of Material Processing Technology, 38, pp. 675-687, 1993.
[10] Ceretti, E., Attanasio, A., Fiorentino, A., Giorleo, L., Giardini, C.: Aluminium can shaping by hydroforming: simulative feasibility study and prototype production, The International Journal of Advanced Manufacturing Technology, 68, pp. 1797-1807, 2013.
[11] Hegadekatte, V., Shi, Y.: Buckling of beverage cans under axial loading, Simulia India Regional Users Meeting, pp. 1-16, 2011.
[12] Sawant, D.A., Venkatesh, M. A.: Buckling and crushing analysis of cylindrical aluminium cans & optimizing the parameters effecting crush strength using FEM, International Research Journal of Engineering and Technology, 3(6), pp. 32081-3085, 2016.
[13] Gönczi, D., Baksa, A., Kiss, L.P.: Alumínium palackok nyakazási lépésinek és stabilitásvesztésének modellezési sajátosságai. Multidiszciplináris tudományok, 9(2),pp. 56-63, 2019. [in Hungarian, Modelling considerations for the numerical simulation of buckling during the necking process of aluminum cans]
[14] Kiss L.P.: The effect of various imperfections ont he buckling of aluminium shells. Acta Technica Corviniensis – Bulletin of Engineering. 13(1), pp. 49-52, 2020.

ACTA TECHNICA CORVINIENSIS – Bulletin of Engineering
ISSN: 2067-3809