14th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2010, Mediterranean Cruise, 11-18 September 2010

FINITE ELEMENT ANALYSIS OF DEEP DRAWING

Dr.Sc. Amra Talić – Čikmiš
Muamer Trako
Mladen Karivan
Faculty of Mechanical Engineering
Zenica
Bosnia and Herzegovina

ABSTRACT
Deep drawing is a process for shaping flat sheets into cup-shaped articles without fracture or excessive localized thinning. The design and control of a deep drawing process depends not only on the workpiece material, but also on the condition at the toolworkpiece interface, the mechanics of plastic deformation and the equipment used.
This paper describes the use of ABAQUS finite element code in a single stage sheet metal forming simulation on rectangle cup deep drawing.
Key words: finite element analysis, simulation, deep drawing

1. INTRODUCTION
In forming processes the final part is exactly defined by dimensions, tolerances and mechanical properties. In order to achieve the production with low costs it is necessary to control the production in every single detail. We need a detailed understanding of the parameters affecting the production process and the final product.
The questions connected to the technology preparation phase of deep drawing process are:
- prediction of fracture,
- prediction of wrinkling prediction of final sheet thickness,
- determination of optimal initial blank geometry,
- prediction of final dimensions of the part (after springback),
- evaluation of loads on the tool.
- prediction of surface deflection.

The most used numerical method for numerical simulation of the forming process is finite elements method (FEM). The numerical simulations included the evaluation of the influence of various factors on the production process, the analysis of various test geometry, as well as the evaluation of loads on the production process. The problems of deep-drawing process are studied on the simple rectangular cup example. The method of using finite element software ABAQUAS CAE.

2. NUMERICAL SIMULATION OF DEEP DRAWING PROCESS
One of the objectives of the FEM simulations was to determine a test geometry both for the FEM model and laboratory tests. In determination of test geometry, the following viewpoints wee considered:
- In order to simplify the tooling system and the test procedure, the geometry should be as simple as possible.

- Its shape should allow the easy strain measurements at any point of the surface.
- During the forming process the friction effect should be minimum.
- Both parts of the FLD (draw and stretch regions) should be covered by the deformation values.
- The results are presented in following sections.

2. 1. Part geometry definition

Here are two types of parts, tools and blank. It is reasonable for tools to be rigid, and only blank has to be deformable. Both tools and blank can be assumed as shells.
Process of modeling is very similar to all CAD softwares (Solid Edge, Solid Works etc.), and it is based on parametric modeling, just as it is in mentioned CAD softwares[1].

2.2. Meshing

Meshing is critical operation regarding the results of numerical analysis, so it has to be done properly and with special care. Here, it is done semiautomatic, radii were divided, as shown in next picture (Figure 1, Figure 2), and all other meshing is done automatically.

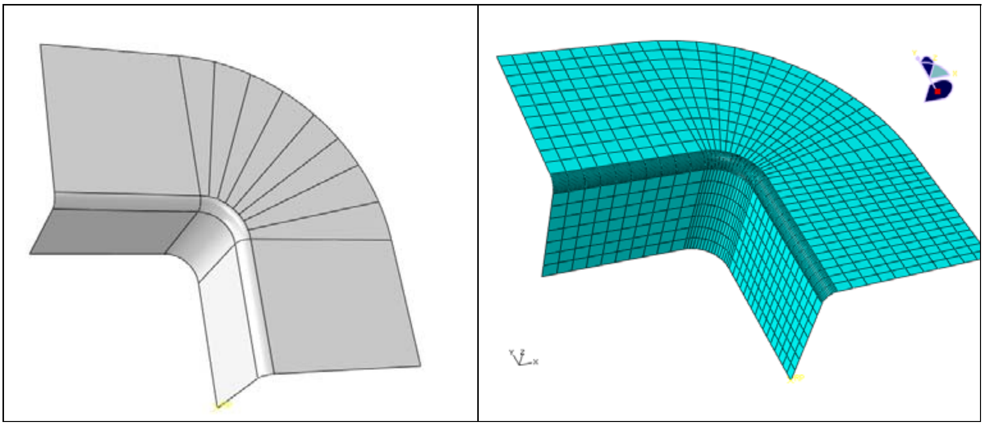

Figure 1. Radius division (die)

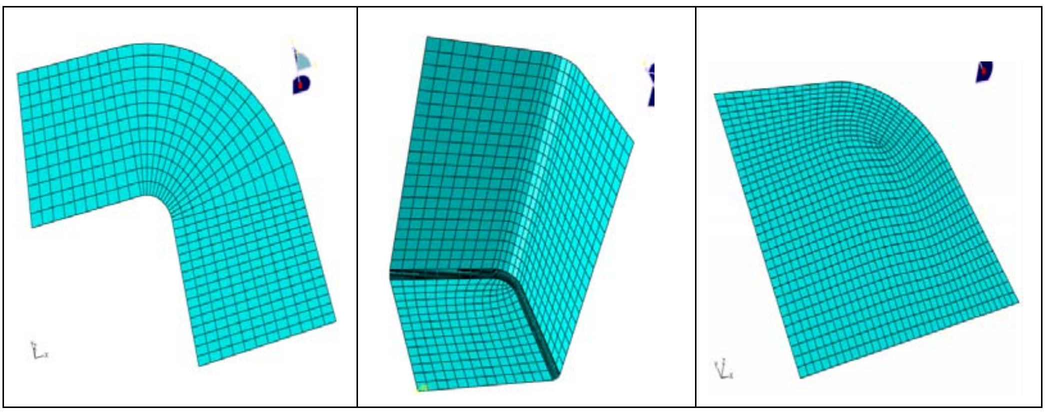

Figure 2. Blank holder, punch and blank meshed

2. 3. Material definition

As mentioned before tools are assumed as rigid, so there is no need to define material, but for blank it has to be defined. It is plastic material. Firstly by experiment, all necessary material parameters are obtained (yield stress – plastic strain curve, elasticity modulus etc.), and those parameters must be imported.

Material is defined according to Krupowsky's law, while flowing curve's coefficients (C,n) and Lankford's coefficients (r_0, r_{45}, r_{90}) for material was used for manufacturing products, were given in table 2.

Table 1. Coefficients which define material features

Coefficients for material St13, and sheet metal thickness s=1.5 mm		
Young′s module E	[N/mm^2]	$2{,}068\cdot10^5$
Poison's coefficient ν	[1]	0,3
ρ	[kg/m^3]	7800
C	[N/mm^2]	555,83
n	[1]	0,214
R_m	[N/mm^2]	322,26
R_p	[N/mm^2]	177,06
$\bar{r}$	[1]	1,68
Δr	[1]	0,71

2.4. Boundary conditions

There are four parts, so every has to be constrained properly. That is why boundary conditions in every FEM analysis are essential. Boundary conditions implementation is divided in two steps (Table 2). In first step constraints are implemented, and in second blank holder force and speed of punch.

Table 2. Boundary conditions

Part	Translations			Rotations			Symmetries		
	X	Y	Z	X	Y	Z	X	Y	Z
Blank									
Die									
Punch			5 m/s						
Blank holder			10 kN						

2.5. Contacts

Contact surfaces used here are:

- top blank – bottom punch
- top blank – bottom blank holder
- bottom blank – top die

2.6. Post processing

Most important results obtained by this kind of simulation are:

- prediction of tearing or evaluation of sheet metal;
- prediction of surface defects;
- prediction of wrinkling sheet metal;
- prediction of finite sheet metal thickness;
- prediction of spring-back and elastic ironing;
- define deformation forces and its distribution on surfaces of tool, etc.

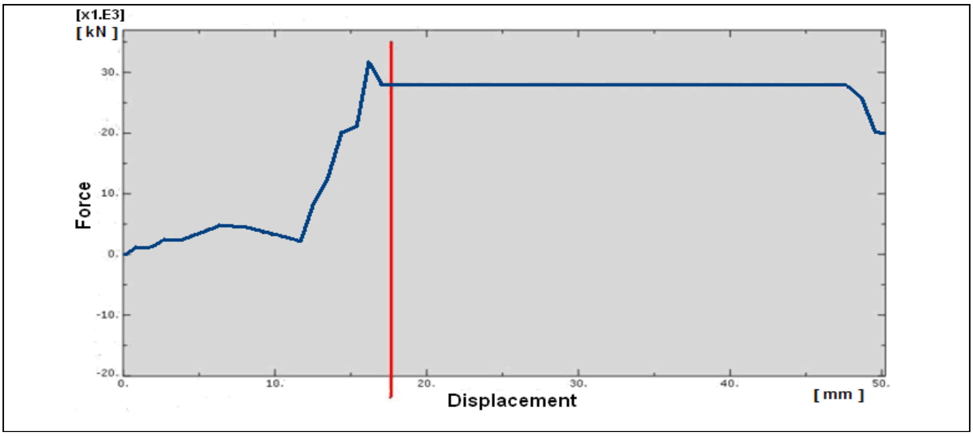

Figure 3. Diagram of deformation forces given during deep – drawing process

As final results of simulation presents deformation forces which have achieved during deep – drawing process (Figure 3.), then finite sheet metal thickness and its distribution on the rectangle part (Figure 4.), stresses (Figure 5.) and deformations obtained for demanded boundary conditions in preprocessing of the performed simulation.

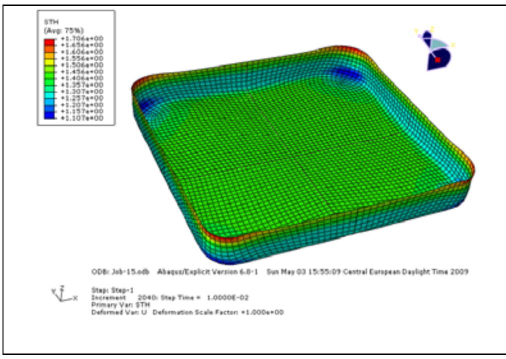

Figure 4. Distribution of sheet metal thickness after finished drawing – operation [2]

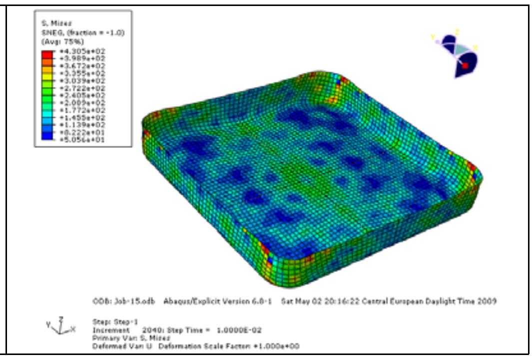

Figure 5. Distribution of stresses after finished drawing – operation (von Misses stress)[2]

3. CONCLUSIONS

The objective of numerical simulation of the forming process is in first order sasses on the market: high quality of product, low – cost, and reduces development time of a new product. In some cases numerical simulations cannot eliminate trial and testing completely and therefore the manufacturing of prototypes and testing tools is required with different rapid prototyping (RP) and rapid tooling (RT) techniques. Numerical solving obtains its value, the most because it offers opportunity using achieved results and a little receding from experimental results.

4. REFERENCES

[1] Abaqus, User Guide

[2] Talić-Čikmiš, A.: Prilog analizi naprezanja i deformacija u procesu izvlačenja pravouglih tijela, doktorska disertacija, 2009.