Study of the influence of punched contour geometry over the punch force using finite element analysis

Stelian Uțuleanu[1, 2,*], *Aurelian* Vlase[1], *Gheorghe* Sindilă[1], and *Nicolae* Căpățână[1]

[1]University POLITEHNICA of Bucharest, Faculty of Engineering and Management of Technological Systems, Splaiul Independenței 313, 060042 Bucharest, Romania
[2]ICPE SA – Special Electric Machines, Splaiul Unirii 313, 030138 Bucharest, Romania

Abstract. When is calculated the punching force is taken into account only the perimeter and not its form. In this case, the pressing force for the contours with the same length, but with different geometries (circle, square, hexagon, triangle and shearing) may have different values. To demonstrate this hypothesis were made experimental tests and they confirm this supposition. Now this hypothesis is verified using the finite element analysis in order to show the possible differences and to view the material behavior during punching process. The analyzed materials are AISI 304, DC01 (AISI 1008), Al99.8 and Brass CDA 110. The software used to make the finite element analysis is Deform 3D version 6.1 (service pack 2).

1 Introduction

Punching is a group of cold plastic processing methods based on wholly or partially material separation, after a closed or open contour [1, 2].

It is used both for obtaining finished parts and semi-finished products for further processing operations by plastic deformation such as bending, drawing, cold forming.

Punching contains several methods like: cutting, punching, shearing, notching, trimming [2, 3].

The software used for finite element analysis (FEA) is Deform 3D Service Pack 2. This system consists of three major components [4]:

1. A *pre-processor* for creating, assembling, or modifying the data required to analyze the simulation, and for generating the required database file.

2. A *simulation engine* for performing the numerical calculations required to analyze the process, and writing the results to the database file. The simulation engine reads the database file, performs the actual solution calculation, and appends the appropriate solution data to the database file. The simulation engine also works seamlessly with the Automatic Mesh Generation (AMG) system to generate a new FEM mesh on the workpiece whenever necessary. While the simulation engine is running, it writes status information, including any error messages, to the message (.MSG) and log (.LOG) files.

* Corresponding author: stelian_u_86@yahoo.com

3. A *post-processor* for reading the database file from the simulation engine and displaying the results graphically and for extracting numerical data.

The analyzed contour length is 60 mm, and the geometries are presented in the next figures (Fig. 1 to Fig. 5).

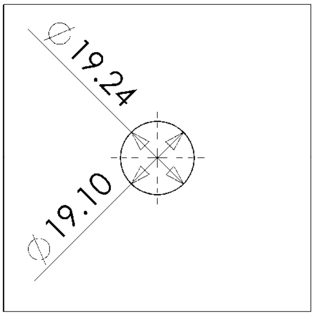

Fig. 1. Circle contour.

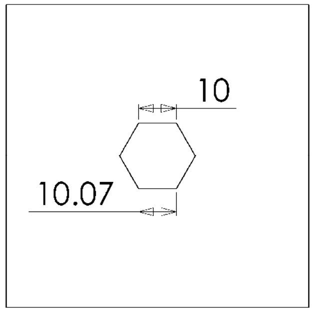

Fig. 2. Hexagon contour.

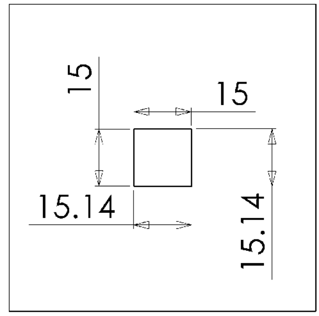

Fig. 3. Square contour.

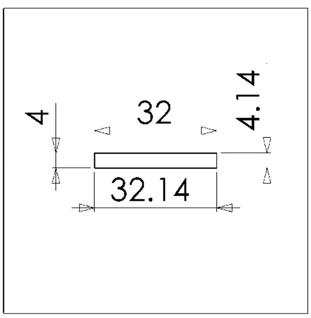

Fig. 4. Rectangular contour.

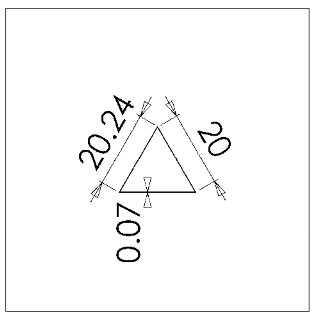

Fig. 5. Triangle contour.

2 Setup

To set up a new simulation it must be selected the “New Problem” under the “File” menu. In the wizard window that appears is selected “Forming” as guided template and after that should be chosen the problem location and its name. This window will be automatically closed and will be opened the pre-processor component.

After is selected the unit system [System International (SI)], should be entered a name for operation, followed by “Cold forming” process type, “Shape Complexity” as “Complex” and “Accuracy” as “Accurate”, “Workpiece Shape” as “Whole part” and “Number of Objects” as “1 workpiece + 2 dies”.

Now can be started the defining of objects: workpiece, top die and bottom die.

These elements were created in SolidWorks 2010 x64 Edition and exported to STL format in order to be imported in Deform 3D.

The first object defined is the workpiece and only it will have a mesh network, because the dies are considered as rigid elements. Can be let the default name for workpiece or can be choose a new one. To add the geometry in the pre-processor can be selected one option from: “Import geometry”, “Extract from mesh” and “Define primitive geometry”.

After is added the geometry of workpiece should be set the number of mesh elements. In our case was selected the maximum number of 100.000 elements, and the “Size ratio” as 3, followed by the mesh generation, the selection of material and the boundary conditions as velocity 0 mm / sec. The materials used were imported from the library (AISI 304, AISI 1008, Al99.8 and Brass CDA 110).

In the next step will be imported the top die and will be set the movement for "Press / Hammer" on Z axis for a mechanical machine with 8 mm of "Total stroke" and with a speed of 2 Stroke/sec.

Also, will be inserted the bottom die and all of these objects will be positioning manually. Friction coefficient will be let as default (0.08), the "Total primary die travel" will be set as 8 mm and will be not checked any "Stopping Controls".

After that will be define the number of steps (400) and step increment (10), followed by the checking of data, generation of database, switching to simulation engine and with starting of this process.

3 Results

After the FEA studies had been finished were created comparative graphs for each type of material, with all five types of punched geometry.

Considering that the top die is 4 mm above the bottom die, the registration of force will start at 3.19 mm stroke for brass, at 3.49 mm stroke for AISI 304 and AISI 1008, respectively at 3.68 mm stroke for Al99.8. The first registered value is when the top die hits the material, and the maximum force is registered when the material is completely ruptured. To highlight the interval where the force presents variation, there was fixed the interval of stroke between 3 and 6 mm. The following figures show these graphs.

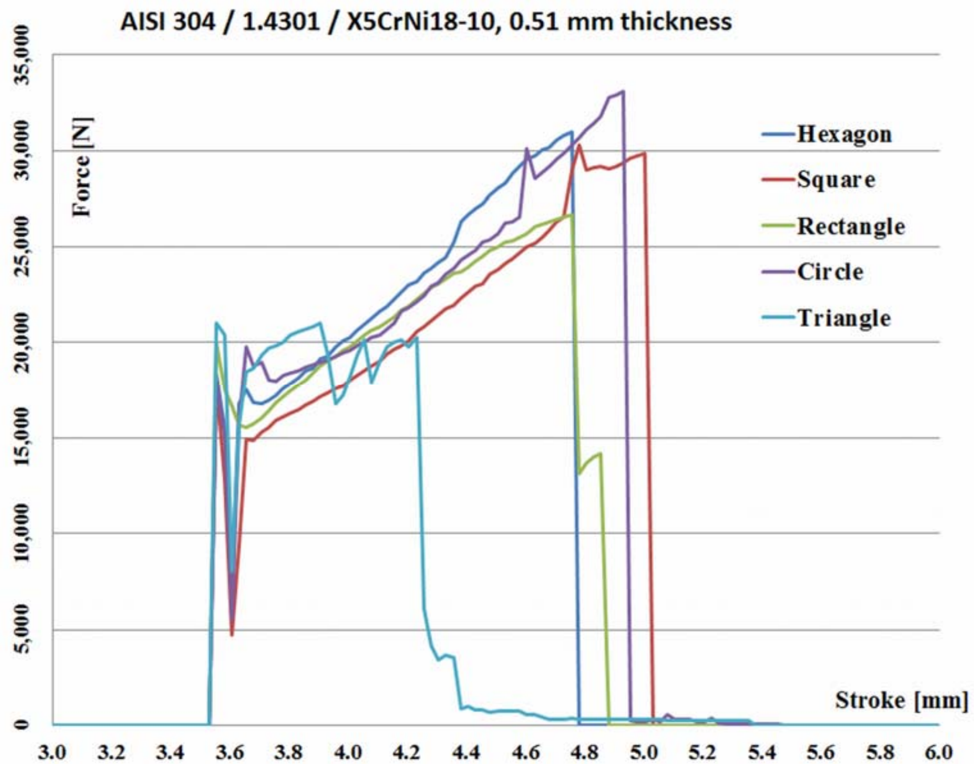

Fig. 6. FEA results for AISI 304 material.

The graphs for DC01 and Al99.8 materials are shown in the figures 7 and 8.

 DOI: 10.1051/matecconf/201711202007

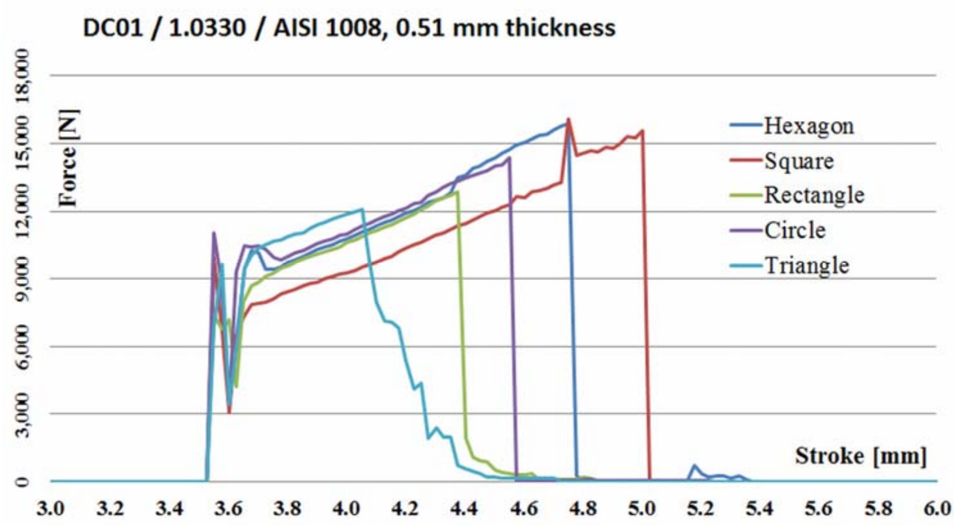

Fig. 7. FEA results for AISI 1008 material.

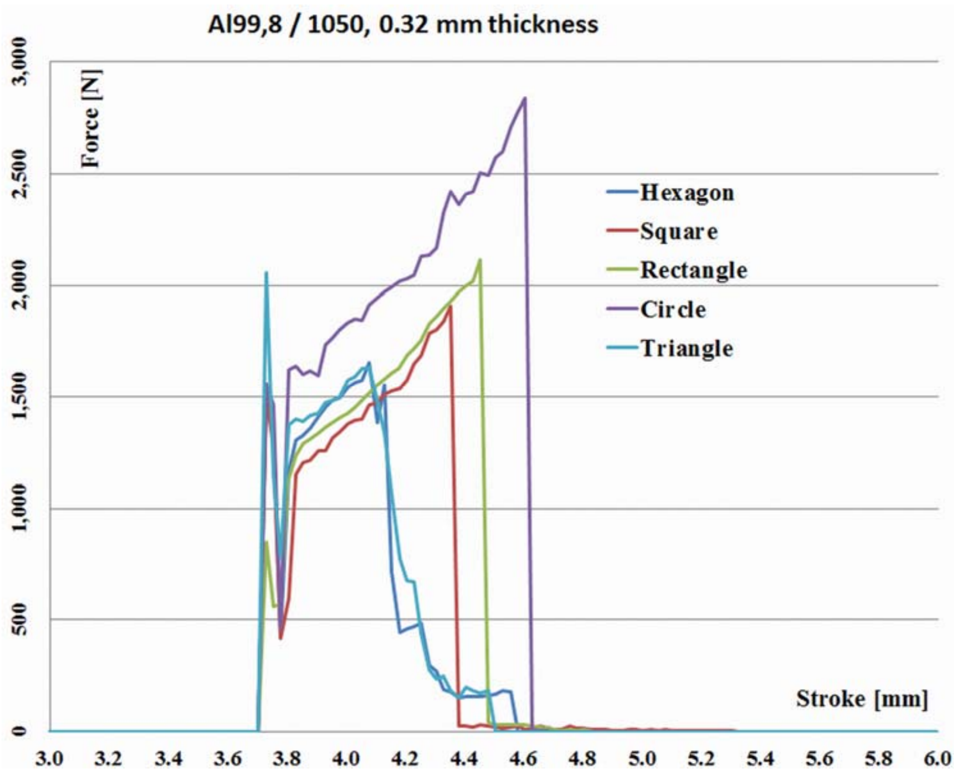

Fig. 8. FEA results for Al99.8 material.

As we can see in these graphs, the geometry of punched contour influence the punching force because the material behavior is different for each type of geometry and also the flow of it is modified with impact over the punching force and over the interval of stroke needed to achieve the full rupture of material (when the force registration is stopped).

The graphs for Brass CDA 110 are shown in the Figure 9.

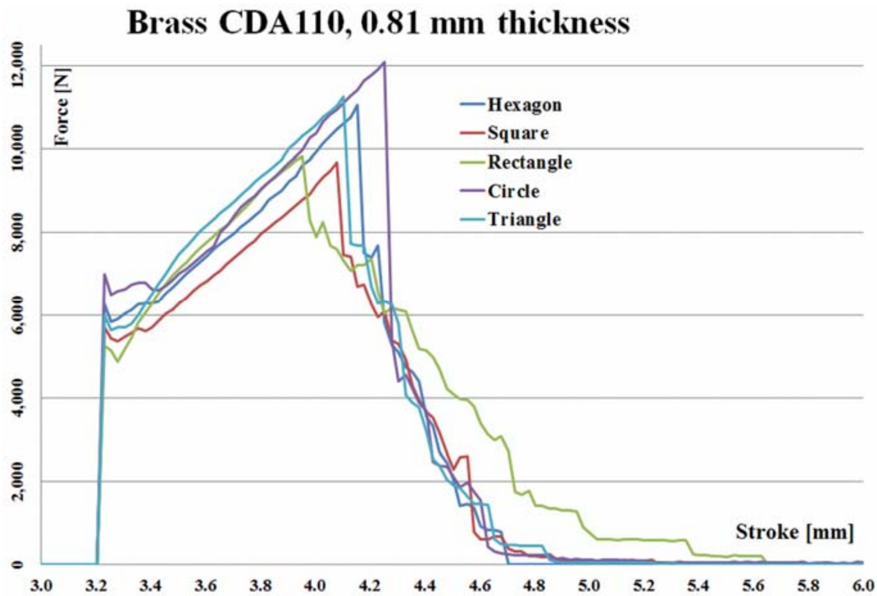

Fig. 9. FEA results for Brass CDA 110 material.

To easily observe the differences between the forces obtained for each type of contour shapes and materials was made a comparative table and was determined the percentage difference between the maximum force and the minimum force.

Table 1. Comparison of influence of punched contour geometry over the punch force.

		AISI 304	AISI 1008	Al99.8	Brass
Hexagon	**F con**	18,366.31	9,408.24	1,540.58	6,269.08
	F max	30,944.10	15,870.65	1,654.75	11,050.73
Square	**F con**	17,945.79	9,938.70	1,506.11	5,712.40
	F max	30,295.54	16,108.86	1,910.74	9,693.75
Shearing / cutback	**F con**	19,980.57	7,416.46	853.24	5,253.49
	F max	26,666.41	12,880.79	2,115.74	9,826.86
Circle	**F con**	17,917.21	11,029.96	1,561.13	6,999.85
	F max	33,089.05	14,386.40	2,838.76	12,095.67
Triangle	**F con**	21,030.08	6,854.26	2,058.01	6,002.73
	F max	21,030.08	12,089.80	2,058.01	11,261.86
Max – min [%]	**F con**	14.80	37.86	58.54	24.95
	F max	36.44	24.95	41.71	19.86

The “F con” represent the force recorded when the top die hit the workpiece and the “F max” represent the maximum force recorded during simulation. Because it is hard to observe in images the differences of material deformation for different geometry contours, in figures 10 and 11 are presented as examples two steps of deformation for AISI 304 material, for circle geometry.

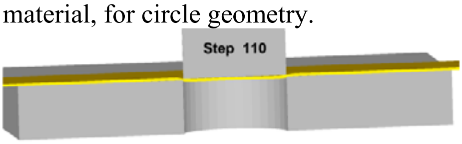

Fig. 10. AISI 304 material deformation at step 110 for circle geometry, 0.51 mm thickness.

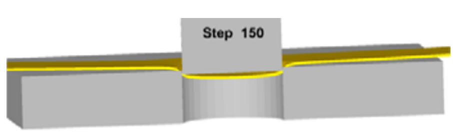

Fig. 11. AISI 304 material deformation at step 150 for circle geometry, 0.51 mm thickness.

DOI: 10.1051/matecconf/201711202007

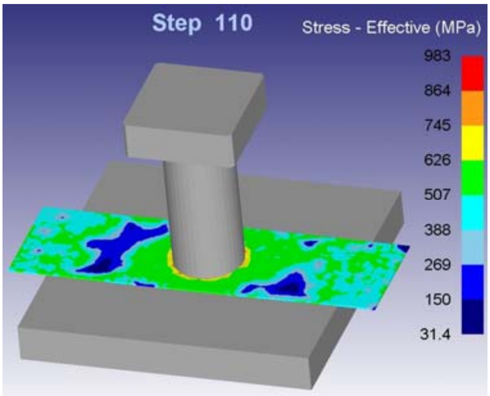

Fig. 12. Stress representation at step 110 for round geometry on AISI 304 material.

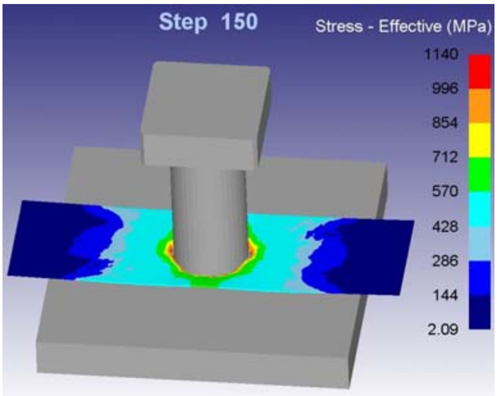

Fig. 13. Stress representation at step 150 for round geometry on AISI 304 material.

Stress representation is more interesting, but due to the limitations to 6 pages of paper, in the figures 12 and 13 were presented the stress analyses at step 110 and 150 for AISI 304.

4 Conclusion

Observing the variations of punched force in the graphs presented in figures 6 – 9 and after the comparison of forces centralized in table 1, we can conclude that our hypothesis about the influence of punched contour geometry over the punch force is confirmed and gives us the opportunity to continue the research.

In further researches will be analyzed the influence of different thickness of material over the punched force, over the material deformation and its flow. Also, there will be used other setups (such as modification of: shape complexity and accuracy, number of elements, minimum element size and size ratio of elements).

Also, will be made a comparison between calculated, experimental and FEA punch forces for the same length of punched contours but with different geometry.

To achieve FEA results very close to the real deformation of material, must be correlated the experimental tests with simulation results and to determine the optimal setup for this kind of process.

References

1. Gh. Sindilă, Cold plastic deformation manufacturing technologies, Ed. Bren, Bucharest (2007)
2. S. Uțuleanu, A. Vlase, Gh. Sindilă, N. Căpățână, ISCAME 1, **212**, Debrecen (2013)
3. C. Ciocârdia, Cold pressing technology, Ed. Did. and Ped., R.A., Bucharest (1991)
4. *DEFORM 3D Version 6.1(sp2) User's Manual,* SFTC , Columbus, Ohio (2008)