Modeling And Simulation Of Combined Extrusion For Spark Plug Body Parts

T.Canta, D. Noveanu, D. Frunza

Department of Materials Processing, Technical University of Cluj, 103 Muncii Avenue,400641 Cluj, Romania

Abstract. The paper presents the modeling and simulation for the extrusion technology of a new type of spark plug body for Dacia Supernova car. This technology was simulated using the finite elements modeling and analysis SuperForm software, designed for the simulation of plastic deformation processes. There is also presented a comparison between the results of the simulation and the industrial results.

INTRODUCTION

The producers of spark plugs for car engines obtain the body part by extrusion technology using 5-6 stages for deformation and sometime the part, is rotated between two stages to ensure a big strain at both ends. A Romanian company produced by cold extrusion, a wide range of spark plug bodies, all having the maximum transversal dimension at one of the ends of the part. The modern engines use a type of spark plugs, which have the maximum transversal dimension in the medium zone of part height.

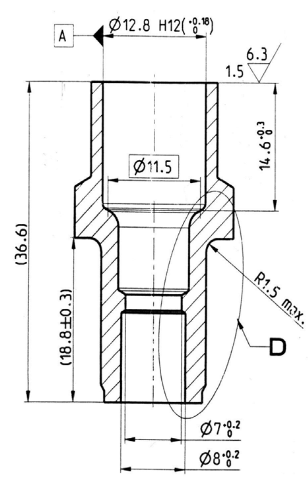

FIGURE 1. The spark plug body

The equipment used for that, allow four stages of deformation (the first stage is dedicated for cutting the bar and the last stage is needed for piercing the part). The challenge was to design the extrusion technology which fit their needs by modeling and simulation using a commercial software MSC SuperForm[1].

FEM SIMULATION

In order to simulate the cold extrusion of the spark plug body there were performed the following steps:

1.Establish the material behavior during cold deformation – obtaining the real σ-ε curves;

2.Determining the contact conditions;

3.Introduction of the geometry for the extrusion tools and for the initial billet;

4.Introduction of material data, friction condition and equipment data;

5.Analysis;

CP712, *Materials Processing and Design: Modeling, Simulation and Applications,* NUMIFORM 2004,
edited by S. Ghosh, J. C. Castro, and J. K. Lee

6.Results interpretation.

EXPERIMENTS

Material Behavior

The material used for this kind of parts is S 18 A, a low carbon steel, designed to be used in cold plastic deformation technologies. The chemical composition is: Cmax=0.18; Mn=0.25 ... 0.50; Si_{max}=0.10; P_{max}=0.025; S_{max}=0.030; Al=0.02 ... 0.07; Ni_{max}=0.20; Cr_{max}=0.20; Cu_{max}=0.20.

There was obtained the real σ-ε curve. The material data needed for simulation, were obtained from cold torsion test, and by using a data acquisition system there was obtained the τ-γ curve. Considering as valid one of the three yielding criteria: Tresca, Von Mises and their average there was plotted the real σ-ε curve as in the next figure:

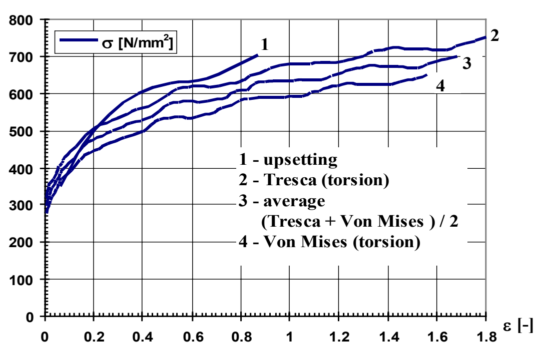

FIGURE 2. The real σ-ε curve[2]

For comparison it was also plotted the real σ-ε curve from a cold upsetting test.

Contact conditions

The friction coefficient can be determined in different ways: the most used method is the ring-upsetting test [3]. For comparison it can be used an alternative method i.e. based on the Ekelund equation[4]:

$$p = 2K\left(1 + \frac{1}{3}\mu\frac{d}{h}\right). \qquad (1)$$

Starting from relation (1) it can be determined the friction coefficient as:

$$\mu = \frac{3(p_2 - p_1)}{(p_1 d_2 / h_2 - p_2 d_1 / h_1)} \qquad (2)$$

where:

p_1 - average pressure of sample 1
p_2 - average pressure of sample 2
d_1 - diameter of sample 1
d_2 - diameter of sample 2
h_1 - height of sample 1
h_2 - height of sample 2

There were pressed cylindrical samples Φ15 x 10 and Φ18 x 10. The axial pressure was continuously recorded via a data acquisition system, and by using equation (2) resulted the continuous variation of the friction coefficient versus strain, presented in Figure 2.

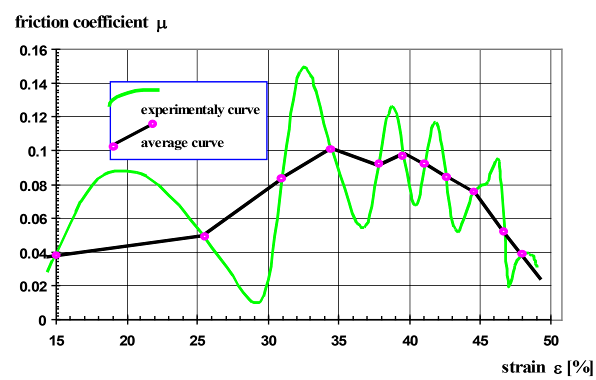

FIGURE 3. Determination of friction coefficient

Finite elements analysis

Pre-processing

The shape of the dies and the punches and of the initial billet was realized using the SuperForm pre-processor. They are presented in Figure 4.

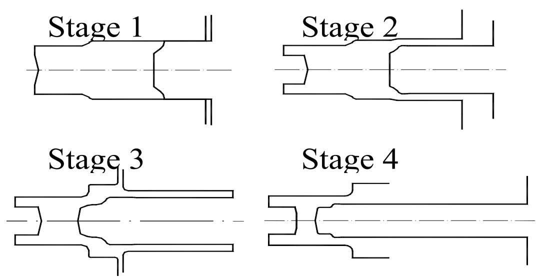

FIGURE 4. Shape of the dies and punches

The equipment data were provided by the Romanian company, which have a multi stage 2.8 MN crank shaft press type Peltzer.

Analysis

The diameter of the billet is near hexagonal size (15.6 mm); this fact leads to smaller strains. In this technology, only the first stage occurs in a closed die (Stage 1); the other stages occur in open dies. So, the stresses in tools are smaller and there is not necessary to cut very accurate the material.

The hexagonal zone is formed inside the die in stage 2; this fact ensures very good conditions for deformation. The flange forms in stage 3 by upsetting the zone under the hexagon. It must be mentioned that in this stage the hexagonal zone is not deformed in the punch. It is only driven in the tool. In the third stage the punch is not driven in die; it stops 1 mm before the die, and the flow was directed to avoid any penetration of the material in the slot. The last stage of deformation achieves the final dimensions of the part, by backward extrusion.

Considering axi-symmetricity, just one half of the part is analyzed [5]. The element used is quadrilateral with 4 nodes. Totally there are 276 elements. Young's modulus was assumed to be 2.1×10^5 MPa and Poisson's ratio to be 0.3.

The initial yield stress is 298 MPa. Analysis with work hardening was performed. The hardening curve which was introduced shown in Figure 5.

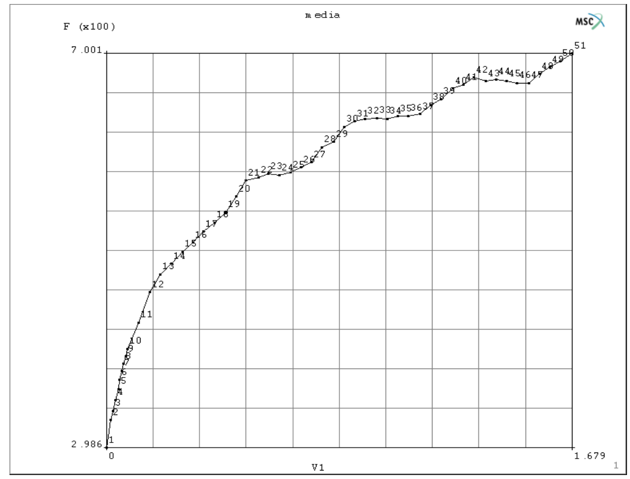

FIGURE 5. Strain-hardening curve

The friction coefficient used for the contact surfaces of the punches and dies is 0.1, the Coulomb principle being adopted.

The whole analysis process is divided into 318 steps, in 4 stages.

The next figures shows the shape of the billet as well as the punch and die geometry at the end of each stage. The plotted parameter is total equivalent stress.

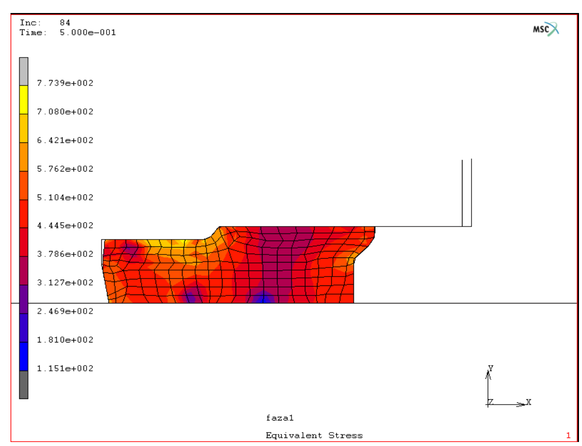

FIGURE 6. Results of the simulation at the end of stage 1

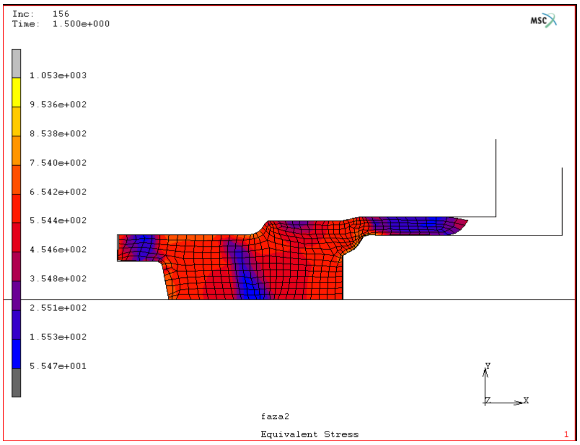

FIGURE 7. Results of the simulation at the end of stage 2

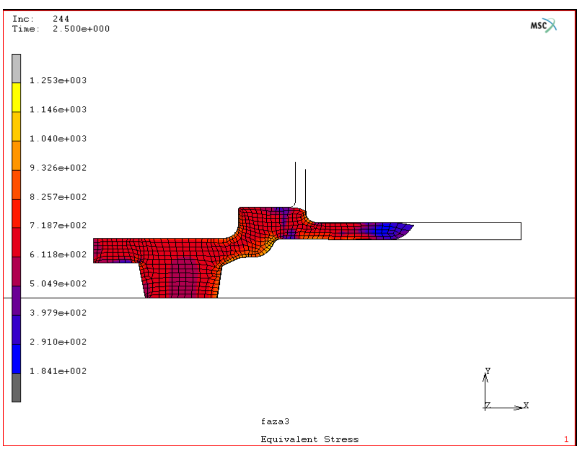

FIGURE 8. Results of the simulation at the end of stage 3

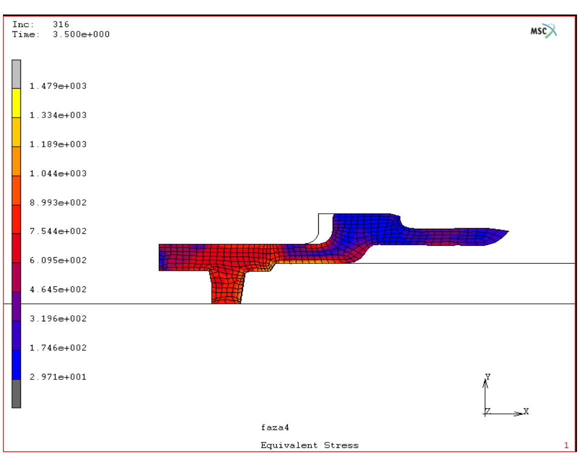

FIGURE 9. Results of the simulation at the end of stage 4

It is important to use a punch with an optimal shape. So, there were studied 8 types of punches. The stresses distribution on their active zone was simulated. It was chosen the punch that achieves the most uniform distribution of stresses.

The final shape of the spark-plug body with the finite elements obtained at the end of the simulation is shown in figure 10.

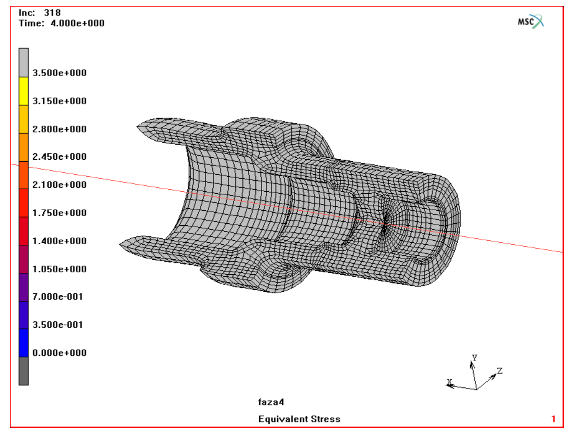

FIGURE 10. Final shape of the spark plug body

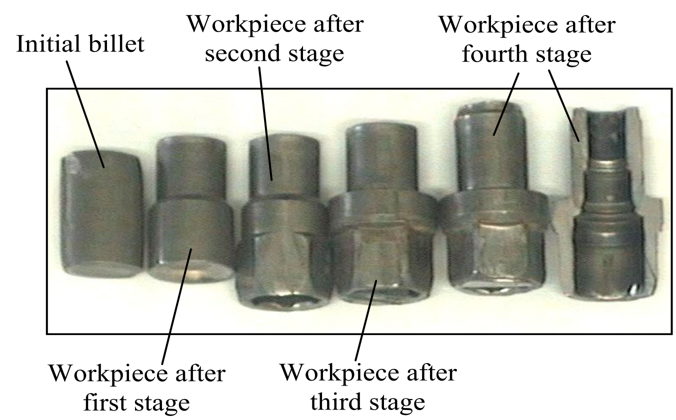

FIGURE 11. The workpiece after each stage

FIGURE 12. Half of the final part

CONCLUSIONS

The results of the simulation agree well with those of industrial process.

1. The loads on the punches at the end of the four stages obtained by simulation are close to those recorded in industrial trials using a data acquisition system.

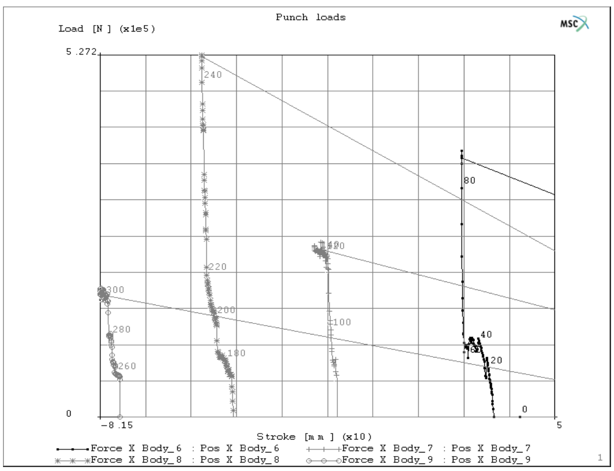

FIGURE 13. Plot of the loads on the punches

where Body_6 represents punch 1, Body_7 represents punch 2, Body_8 represents punch 3, Body_9 represents punch 4,

Table 1. Comparison Between Loads Obtained by FEM Simulation and by Industrial Trials

	Stage 4	Stage 3	Stage 2	Stage 1
Load (FEM) [kN]	175.3	527.2	254.5	230.4
Load (Ind.) [kN]	177	537.7	260.8	239.6

These small differences occur due to the material properties and to the friction coefficient in industry trials.

2. Concerning the shape and dimensions, there are also very small differences between FEM simulation and industrial trials.

Table 2. Comparison Between Dimensions Obtained by FEM Simulation and by Industrial Trials

Length of the part (FEM)	Length of the part (Ind.)	Diameter of the part (FEM)	Diameter of the part (Ind.)
36.38 mm	36.35 mm	13.08 mm	13.05 mm

The differences are due to the elastic deformation of the press and dies, and also as a result of different height of the initial billet resulted after shearing of the initial billet.

REFERENCES

1. Sabadus, D., Frunza, D., Noveanu, D., Canta, T., "Simulation and Modeling for Complex Extrusion of New Spark Plugs Body Parts", Acta Technica Napocensis, vol. 45, Part 1, Cluj Napoca, 2002, pp. 375-380.

2. Tintelecan, C., Canta, T. "Finite Element Analysis of the Complex Shape Workpieces", Proc. of Int. Conference MATEHN'98, Cluj Napoca, 1998, pp. 363-368

3. Male, A.T., and Crockcroft, M. G. "A Method for the Determination of the Coefficient of Friction of Metals under Conditions of Bulk Plastic Deformation" *Journal of the Insitute of Metals,* vol. **93**, pp. 38-46, 1964-65.

4. Ekelund, S., *Steel,* **93** (Aug.21), 1933, pp. 27-29.

5. MARC/Superform User's Guide.