A R C H I V E S O F M E T A L L U R G Y A N D M A T E R I A L S
Volume 57 2012 Issue 4
DOI: 10.2478/v10172-012-0132-2

D. WOŹNIAK*, M. GŁOWACKI*, M. HOJNY*, T. PIEJA**

APPLICATION OF CAE SYSTEMS IN FORMING OF DRAWPIECES WITH USE RUBBER-PAD FORMING PROCESSES

ZASTOSOWANIE SYSTEMÓW CAE W PROJEKTOWANIU PROCESÓW TŁOCZENIA Z UŻYCIEM ODKSZTAŁCALNYCH NARZĘDZI

This article shows example result of computer simulations supporting production process of bearing housing of aircraft engine. Verification of both deep drawing process project and tools design were carried out using finite element models implemented in eta/Dynaform 5.8.1 system and LS-DYNA solver. Wrinkling and fracture of the material were the main phenomena subjected to the investigation on the way of numerical analysis. A number of computer simulations were carried out in aim to analyze the deformation and strain distribution in the final product, as well as to eliminate the mentioned defects. In addition the comparison of results of both industrial tests and computer simulation was done.
Keywords: drawing, rubber forming, FEM modeling

W artykule przedstawiono przykładowe wyniki symulacji komputerowych wspomagających proces produkcyjny elementu silnika lotniczego z wykorzystaniem narzędzi elastycznych. Projekt narzędzi oraz ich weryfikację do procesu tłocznia przeprowadzono z wykorzystaniem systemu eta/Dynaform 5.8.1 i jego solvera LS-DYNA. Pofałdowanie i zrywanie materiału wytłoczki to typowe trudności jakie napotkano w trakcie analiz numerycznych. Przeprowadzono szereg symulacji komputerowych mających na celu wyeliminowanie pojawiających się typowych dla procesu tłoczenia wad. W ramach prac przeprowadzono także próby przemysłowe. Wykorzystując nowoczesne techniki pomiarów, wyniki ze wstępnych prób przemysłowych, porównano z wynikami symulacji komputerowych.

1. Introduction

Permanent technological progress and still growing industrial competition bring a lot of changes in drawing technology of metals. Hydroforming and flexible forming technologies become more and more popular. The second of them is often used in small-series production, mainly in aircraft industry. Forming process with use of flexible tools has a lot of advantages, mainly tools profitability and production flexibility. The flexible tools can give a variety of final product shapes, which sometimes can be very complicated. The idea of classical rubber forming process is shown in Fig. 1. A rubber pad is placed in steel or cast box, which provides the maintenance of pressure exerted by the rubber. Rubber pads are usually made as a monolithic or layered units. The first one is made by vulcanization several 25 mm thick rubber sheets in one block of expected thickness. Second kind of pads applicable in forming processes is made by placement of the rubber sheets one upon the other [1]. In that case they are not vulcanized. Longer life of rubber pad scan be achieved by application of round corner punches, which also are used in the process. It is important to remember, that rounding radiuses should provide an expected shape of the product.

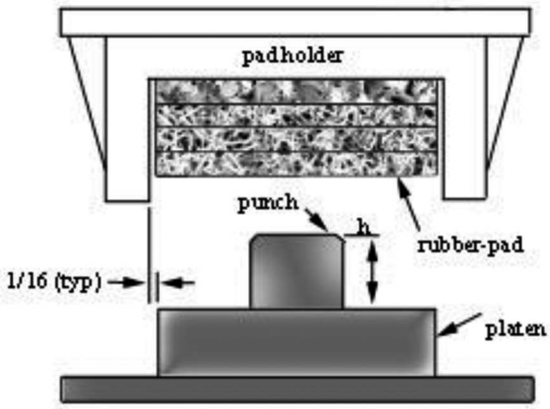

Fig. 1. The Guerin process

* AGH UNIVERSITY OF SCIENCE AND TECHNOLOGY, 30-059 KRAKÓW, 30 MICKIEWICZA AV., POLAND
** WSK "PZL-RZESZÓW" S.A., 35-078 RZESZÓW, 120 HETMAŃSKA STR., POLAND

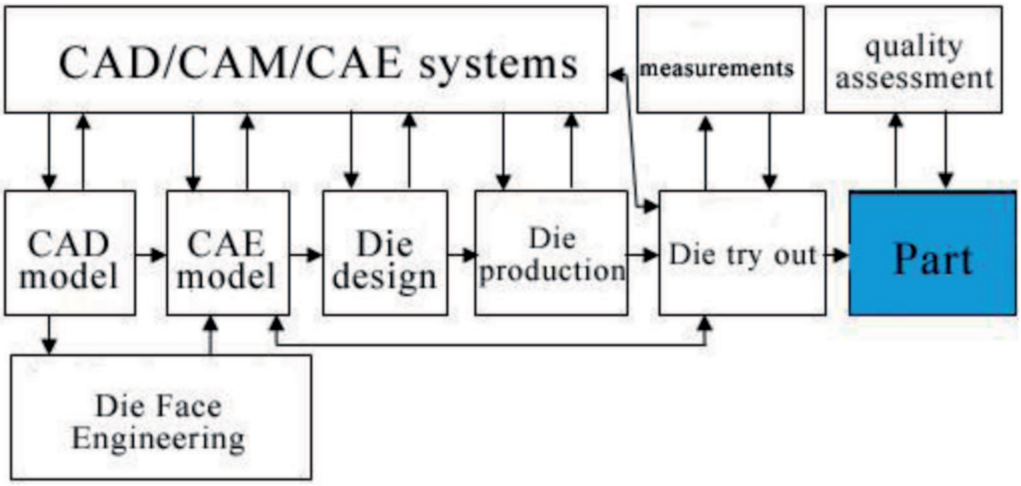

Fig. 2. A product development process with FE simulation support

Aid of advanced computer methods supporting the production processes become more and more popular in the field of development of new forming technologies. Computer Aided Engineering (CAE system) is recently the most popular technic supporting the technological design. One of the main profits coming from such systems are the lowering of project and production costs due to application of fast and accurate design with the help of computer systems. The systems can help not only in development of a new product technology but also in production processes. The simulation outcome scan constantly be updated which results in process optimization.

Computer simulations and analysis of its results can be useful at any stage of metal forming processes. It can be used to analysis of many process parameters like material flow, effective stresses, equivalent plastic strains, deformation, thickness distribution, temperature distribution and first of all for predictive analysis of defects and damage [2]. Today the FEM systems became a part of production cycle, which should include design, production and final product optimization process (Fig. 2).

Decisions made at the design stage of technological process influences both its later realization and the total production costs. There are a lot of commercial packages which are used to simulations of metal forming. In the field of drawing the eta/ Dynaform system and LS-DYNA solver are the leaders. Since 1993 solver of LS-DYNA system is widely used for simulation of deep drawing. On the other hand the Eta/Dynaform system contains dedicated: die face engineering (DFE), blank size engineering (BSE), die structural analysis (DSA), springback compensation (SCP) and line die (LSD) modules [3, 4].

The application of CAE, CAD or CAM system in industrial processes allows accurate analysis of production models. Working on deep drawing processes using both classical and rubber pad technology the user has possibility to define tools, shape and material properties as well as optimize the process on every design stage. This is the main advantage of applied computer systems.

2. Numerical modeling

2.1. MES model

The geometrical modeling of the set of tools taking part in the FEM simulation has been developed with the use of CAD software. Main parts of the tool set: punch, ring and screws are presented in Fig. 3. The shape of punch was designed on the base of the final product model. In the deep-drawing process take also part presented in Figure 4 rubber-pad and blank. Rubber pad consists of two stable elements and third one which is placed loosely on the blank.

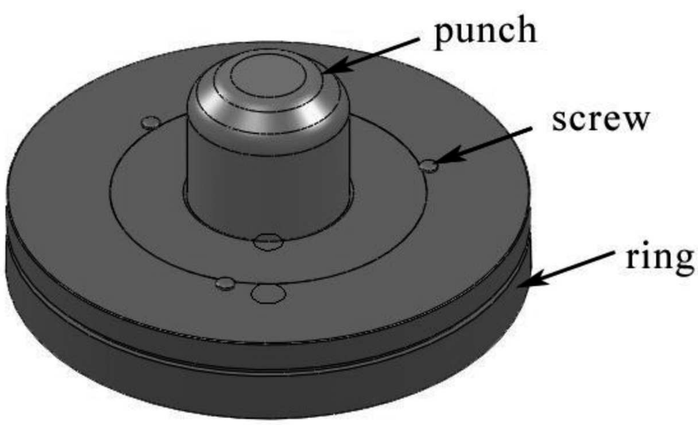

Fig. 3. Stamping tools – CAD model

Fig. 4. Three-layer rubber pad model

After the tool set is prepared the FEM model of deep drawing simulation is ready to simplification. Surfaces which don't take part in the process are deleted. It concerns such surfaces which have undefined contact with the material. Such operation allows simulation time reduction without influence on very good calculation accuracy.

The tooling assemblage which was used in the present study for the simulation of the aircraft engine component stamping process is shown in Figure 5 together with the mesh of finite elements.

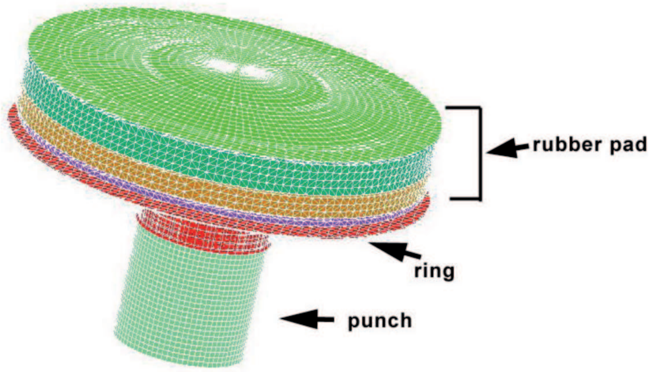

Fig. 5. FEM tools model

The numbers of elements and nodes used for purposes of the current simulation are listed in Table 1.

TABLE 1

Number of elements and nodes

Mesh	*Elements*	*Nodes*
Punch	3195	3206
Ring	3072	2912
Blank	6001	6072
Rubber-pad: first layer	29958	8239
Rubber-pad: second layer	60700	13586
Rubber-pad: third layer	86212	17297
Total	**189138**	**51312**

2.2. Hyperelastic material model

Important parts of the applied FE model are the material models. Material adopted in the current study for blank was a kind of nickel super alloy which in the LS-DYNA database is defined as material model no 37 (Transversely_Anisotropic_Elastic_Plastic). Rubber pad material was defined as hyperelastic. It is characterized by both a very large range of elastic deformation (up to hundreds of percent) and by the incompressibility.

The Mooney-Rivlin model is most widely used to modeling rubber parts. It is multi-parametric model with constant coefficients. The number of parameters used to description of a material model depends on the number of inflections points of the material curve [5]. In the present paper, two-parameter Mooney-Rivlin model was used.

The strain energy density function for an incompressible Mooney-Rivlin material is given by following equation:

$$W = c_{10}\left(\bar{I}_1 - 3\right) + c_{01}\left(\bar{I}_2 - 3\right) \tag{1}$$

where and are the first and the second invariants of the unimodular component of the left Cauchy–Green deformation tensor, and and are empirically determined material constants.

The correct definition of a solid material must ensure a positive value of the strain energy, which leads to the condition:

$$c_{10} + c_{01} > 0 \tag{2}$$

2.3. Results of sample computer simulations

A series of simulation were performed in aim to design the aircraft engine component production process. Mechanical properties of the blank material such as Young modulus, Poisson ratio and stain-stress relationship were results of experimental tests. Each numerical analysis was characterized by appropriate shape of tools and rubber properties. In subsequent numerical experiments the changes of pressure curve and friction coefficient were taken into consideration according to the results of previously done analyses [6]. All these parameters have great impact on the deformation process which is very sensitive even for minimal their changes.

In early stages of the FE analysis a set of tools contained only one layer rubber pad. In further cases two additional layers of rubber were added, as it is shown in Figures 4 and 5. The effect of number of the rubber pad layers and its influence on the stress and strain distribution was also a subject of the analysis. The rubber pad was made of polyurethane having Shore hardness in range 60 to 100 degrees. A number of rubber pads with different initial harnesses have been investigated.

Early results showed an excessive wrinkling which was caused by wrong choice of parameters of the stamping process. Further modifications of the stamping technology has permitted a properly stamped final part, which was free from unwanted defects.

The maximum value of the pressure during the process was 87.5 MPa. Figure 6 shows examples of the pressure curves. Figures 7 and 8 present the thickness distribution of the product in case of application of linear (curve 1 in Fig. 6) and non-linear (curve 2 in Fig. 6) pressure evolution, respectively.

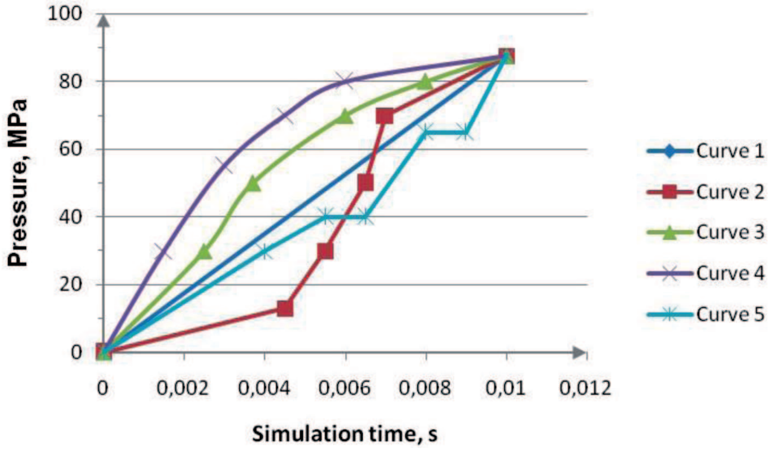

Fig. 6. Examples of the pressure distribution curves during the simulations

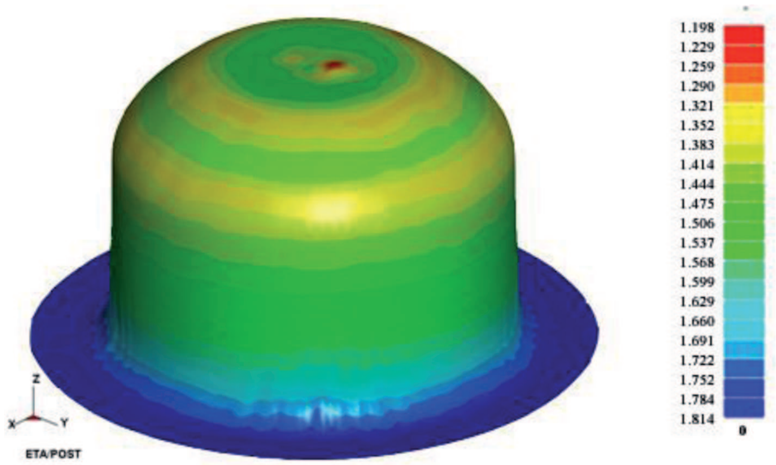

Fig. 7. Thickness distribution for linear increase of pressure

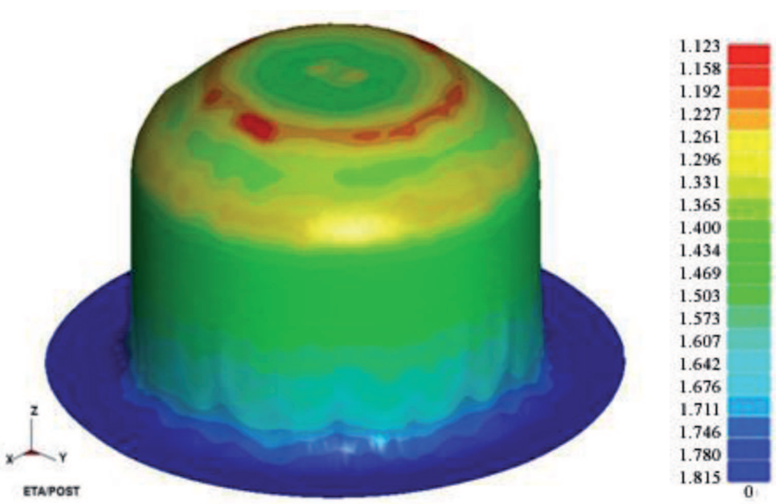

Fig. 8. Thickness distribution for modified pressure increase

The differences in the shape and thickness of the drawpieces are caused by variable pressure values and modification of their trajectory. In both cases, the greatest thickness of the material can be observed on the flange of the product. This area is not considered as an object of the analysis, because it is a waste. Too high pressure or its improper influence on the blank may cause the risk of crack.

Numerical analysis has also shown that even a minimal change in the value of friction coefficient influences the quality of the stamping process. For considered case the friction coefficient has varied in the range between 0.1 and 0.25. Too high values of friction lead to crack and the formation of large flange (Fig. 9).

The final shape of bearing housing is show in Figure 10. It is clearly seen that the final shape is free from typical defects of stamping process. Appropriate modification of stamping technology parameters based on computer simulations allowed to receive the final part which is in accordance with the objectives of the technological process. Analysis of thinning distributions (Figure 11) indicates that the thinning varies from -20.7% to 19.2%.

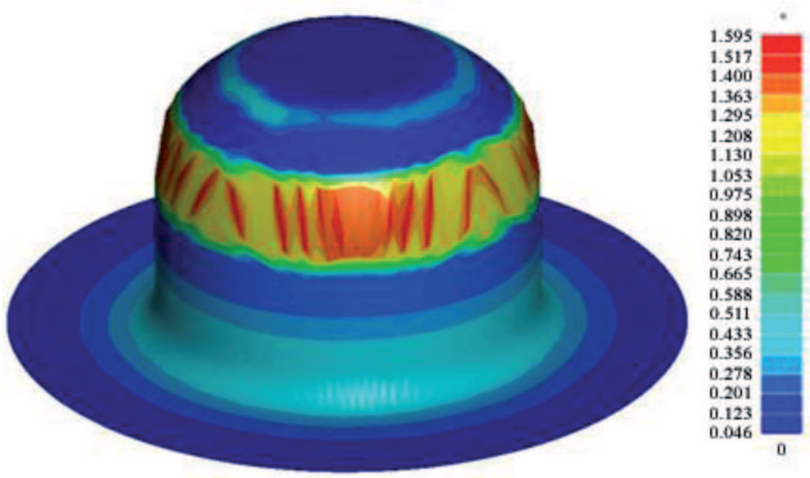

Fig. 9. Major strain for the coefficient of friction of 0.125

Fig. 10. Final shape of bearing housing

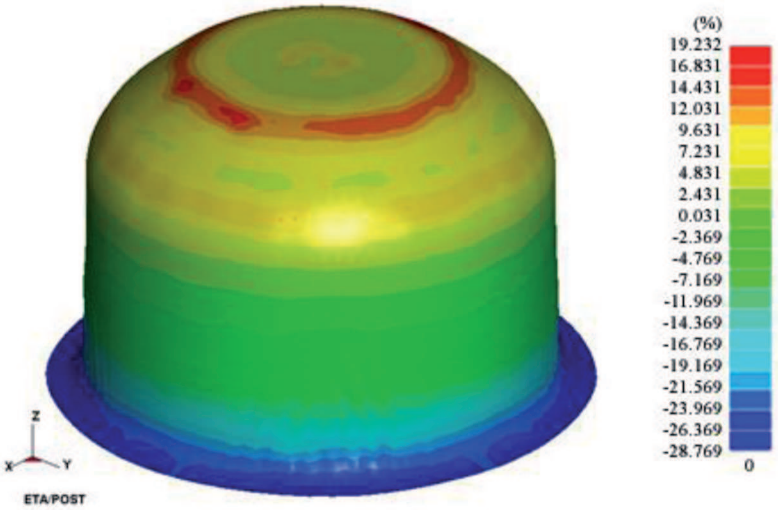

Fig. 11. Thinning distribution for the final shape of bearing housing

3. Industrial tests

The industrial rubber pad forming was conducted using an SAAB 400 press type which is shown in Figure 12.

Fig. 12. SAAB 400

Stamping process was performed using so called punch method which schematically is shown in Figure 13. The forming tools consist of blank holder and one or several punches. The punches are mounted onto a column which passes through the press bed and the tool holder. The tool holder, mounted on the movable press bed contains a blank holder and it is a part of the cushion unit. The forming consists of several successive steps.

The cycle starts with the main ram in its upper position and the sheared blank is placed on the surface of the pressure pad. When the membrane comes in contact with the blank, the forming unit pressure chamber is then closed by the blank holder. The continuing downward motion of the press ram creates a pressure in the forming unit and drawing of blank begins.

When the membrane is in contact with the blank, the chamber of the device is closed by the pressure pad. Further upward movement of the press creates a pressure which starts the sheet metal forming process. This pressure is dependent on the force of resistance, and maybe adjusted during the process. After the membrane reaches the metal sheet and forming chamber is closed the movement of forming unit is temporarily suspended. Such action allows to achieve the correct pressure required for process initiation. After that the process is in progress again. The ability to control the pressure during the forming process makes this technique applicable for the deep-drawing process. Variation in pressure during the process is one of the key parameters. Clever control of this value allows to receive very complicated shapes during the formation with the help of gums. The shaping process proceeds to the point where the pressure pad reaches its lower position, where in most cases an increase of pressure is observed. The last forming stage is the back movement of the lower bed to its initial position. The final product can be easily removed from the punch and the whole process can be repeated.

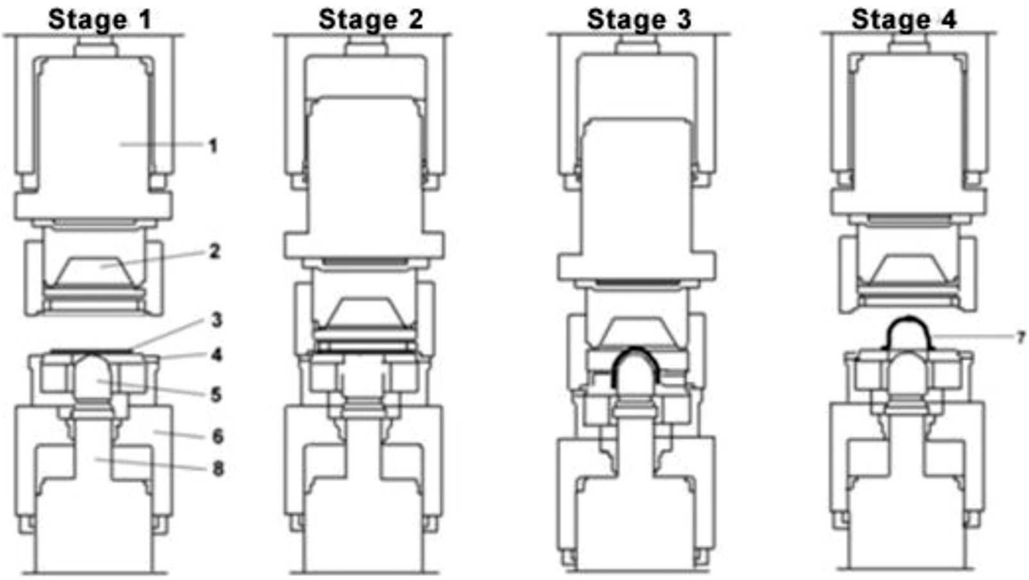

Fig. 13. Forming stages in the punch method: 1. main ram, 2. oil dome, 3. blank, 4. blank holder, 5. punch, 6. lower ram, 7. finished part, 8. column

Fig. 14. The example view of bearing housing before trimming operation

A charge to the stamping process with deformable tools was a round sheared blank having 260 mm in diameter and a thickness of 1.5 mm. It was cut using laser equipment from a metal sheet with dimensions of 1000×265 mm. Admissible thickness variation was ±20%. The final product is shown in Figure 14. According to predictions made with the help of computer simulation the part was free from typical drawbacks of the deep-drawing process, such as folding or material cracks. The resulting shape is in agreement with the theoretical shape. The industrial test results confirm the huge benefits of FEM system for the design and verification of tools for stamping process.

For more detailed verification of computer simulations an analysis of the thickness distribution was done and its results were compared with results of measurement of the real industrial part. Measurements were done along two perpendicular lines shown in Figure 15. The results are shown in Figures 16 and 17.

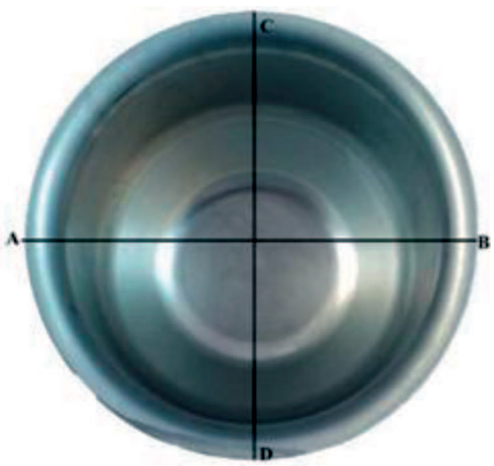

Fig. 15. Vertical sections analysis

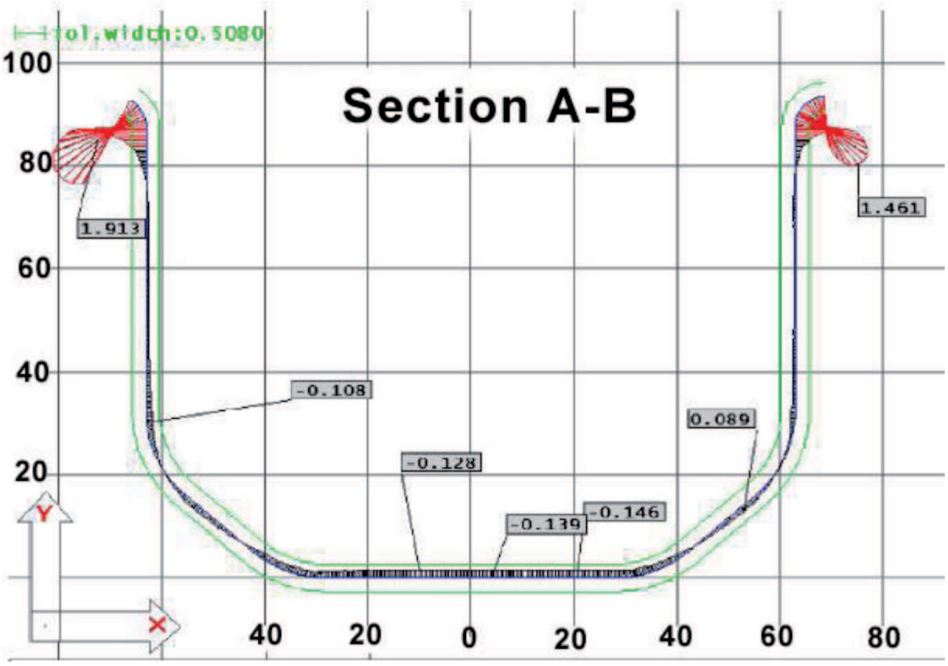

Fig. 16. Thickness distribution for A-B section

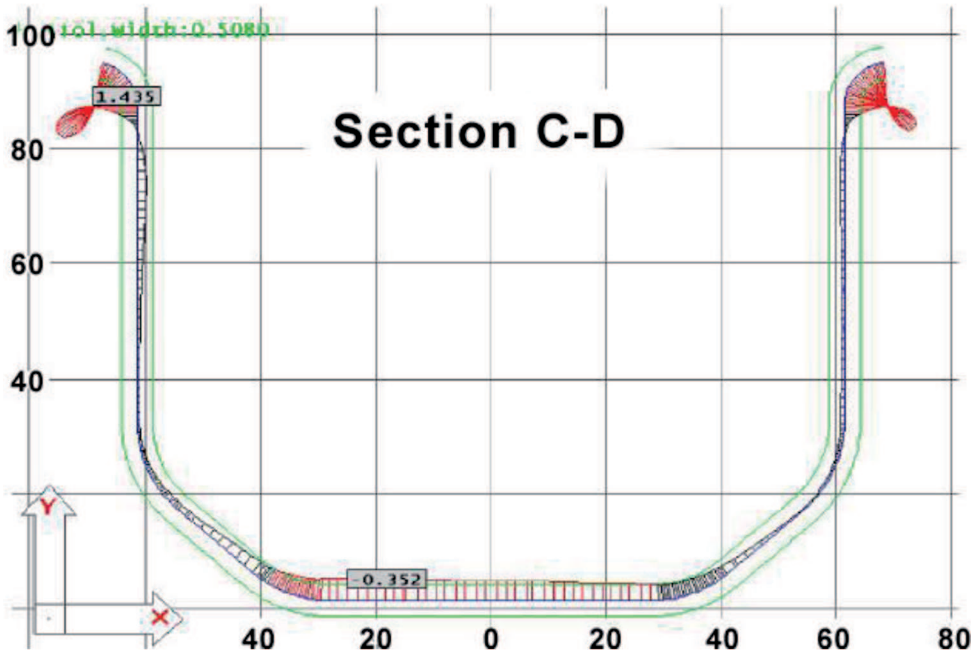

Fig. 17. Thickness distribution for C-D section

The dimensional tolerance of the drawpiece was established. The thickness variation of the final element as compared to the simulation assumptions are small and its maximum values reach 0.352 mm for section A-B and 0.146 mm for section C-D from the Figure 15. Deviation values for the flange are not taken into account because these areas are cut off in trimming operations and are treated as waste. Thickness analysis has shown that its computed distribution is in agreement with results of industrial tests.

However, for more detailed analysis and confirmation of the reproducibility of the results, the distribution of the real drawpiece shape was measured with the use of blue light LED technology. This technique is increasingly being used in industry. Modern scanners allow measurements in any lighting conditions. The reliability and precision of measurement is the main advantage of this technique. Figure 18 shows a color map of the drawpiece shape deviation.

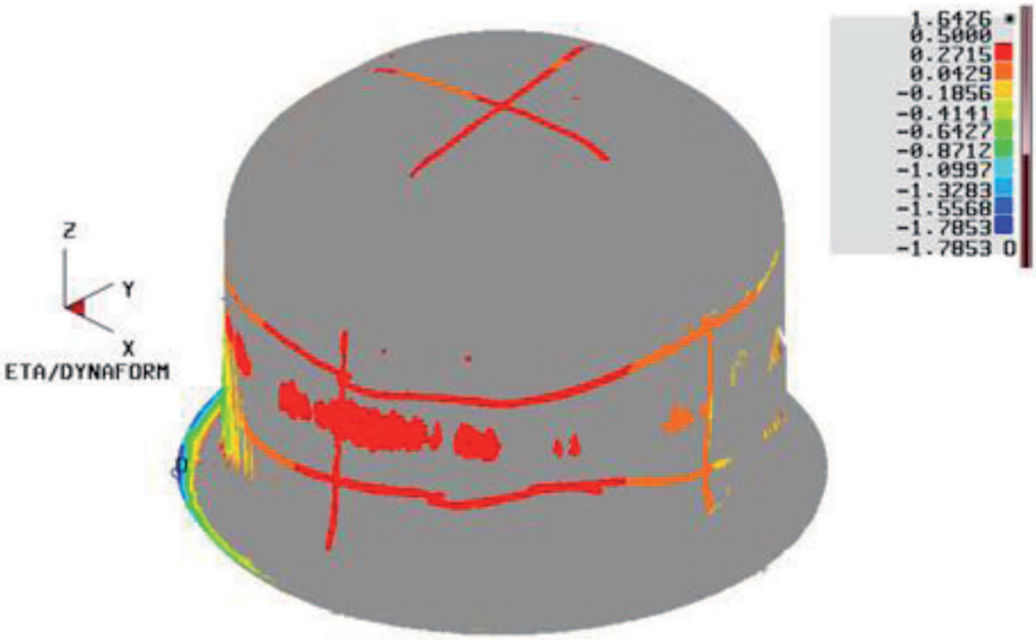

Fig. 18. Color deviation map of the drawpiece

The values of thickness were in the range between -0.4141 mm and 0.2715 mm. They are acceptable and consistent with the results obtained from the computer simulation and production assumptions.

4. Conclusion

In the present work the die face design for stamping of bearing housing of an aircraft engine was investigated using FEM systems. The computer simulations were performed using eta/Dynaform 5.8 system and the LS-DYNA solver. The cause of the formation of wrinkles was studied on the basis of the metal flow obtained from the results of computer simulation of stamping process. The process parameters such as pressure, level of friction coefficient and hardness of the rubber pad were analyzed. The main result of the study is an optimum design of stamping tools. It was performed on the basis of finite element analysis. The industrial tests have confirm achievement of defect free products. The analysis of thickness distribution has shown compatible results coming from both industrial tests and computer simulation.

The work was done with financial support of AGH (grant no 11.11.110.011).

REFERENCES

[1] Engineering Technology Associates, eta/Dynaform 5.8 User's Manual, 2010.

[2] J.O. Hallquist, LS-DYNA Theory manual, California Livermore Software Technology Corporation, (2006).

[3] M. Hojny, Application of an integrated CAD/CAM/CAE/IBC system in the stamping process of a bathtub 1200S, Archives of Metallurgy and Materials **55**, 713 (2010).

[4] M. Hojny, M. Głowacki, A. Opaliński, D. Woźniak, Computer aided design of stamping process technology using the eta/DYNAFORM 5.8 system, Mechanika **7**, 170 (2011).

[5] G. Sala, A numerical and experimental approach to optimize sheet stamping technologies: part II – aluminium alloys rubber-forming, Materials and Design **22**, 299 (2001).

[6] D. Woźniak, M. Hojny, T. Pieja, Computer aided design of stamping technology of aircraft engine element, Proc. Quality of technological processes, Sromowce Niżne, in Polish, 157 (2012).

Received: 10 May 2012.