FINITE ELEMENT EVALUATION OF THE FAILURE IN A DEEP DRAWING DIE USED IN THE MANUFACTURING OF AN AUTOMOTIVE SHOCK ABSORBER ENDCAP

Guilherme C. R. Moura
Federal University of Minas Gerais, Department of Mechanical Engineering, Av. Antônio Carlos 6627 Campus da Pampulha, Belo Horizonte, Minas Gerais, CEP 3270-901

Maria Teresa Paulino Aguilar
Federal University of Minas Gerais, Department of Materials and Civil Engineering, UFMG, Rua Espirito Santo 35, Belo Horizonte, Minas Gerais, CEP 30160-030
teresa@demc.ufmg.br

Haroldo Béria Campos
Federal University of Minas Gerais, Department of Mechanical Engineering, Av. Antônio Carlos 6627 Campus da Pampulha, Belo Horizonte, Minas Gerais, CEP 3270-901
haroldo@demec.ufmg.br

Antônio E. M. Pertence
Federal University of Minas Gerais, Department of Mechanical Engineering, Av. Antônio Carlos 6627 Campus da Pampulha, Belo Horizonte, Minas Gerais, CEP 3270-901
pertence@demec.ufmg.br

Paulo Roberto Cetlin
Federal University of Minas Gerais, Department of Metallurgical and Materials Engineering, UFMG, Rua Espirito Santo 35, Belo Horizonte, Minas Gerais, CEP 30160-030
pcetlin@demet.ufmg.br

Abstract. *A deep drawing die, employed in the manufacturing of an automotive shock absorber endcap, presented a premature wear in a part of the region of contact with the blank. This led to unacceptable scratching and surface finish in the produced endcaps, resulting in an excessive scrap rate and undesirable finishing operations in order to repair the defects, when possible at all. A finite element analysis (FEA) of the deep drawing operation was performed, in order to analyze the causes of the premature wear. Initially, parts produced in a new die and in a worn die were compared. The loads and strains in the endcap and in the dies were then evaluated, through a finite element simulation utilizing the DEFORM 7.0 software. The validation of the simulation was performed through a comparison of the experimental measurements of the thickness of the part with those supplied by the numerical simulation. The validated FEA model indicated a high radial stress in the die, which was close to the acceptable level for the die material. Two different geometries of the punch were then analyzed, involving an increased gap between the die and the punch at the moment when high stresses were observed in the die. This allowed a significant reduction in die loading, without loss of final product geometry.*

Keywords*. Finite Element, Metal Forming, Shock Absorber Endcap*

1. Introduction

The automotive and car parts industries have been deeply involved in productivity increases and cost reductions in their activities. An important aspect of this strategy is the replacement of machining by forming in the manufacturing processes. Figure (1) illustrates a typical car shock absorber. The endcap (1), which is discussed in this paper, is welded to a tubular body (2) and to the fastening ring (3) of the shock absorber. This ring is screwed to the car body, allowing the shock absorber to actuate during the oscillations and impacts undergone by the car wheels.

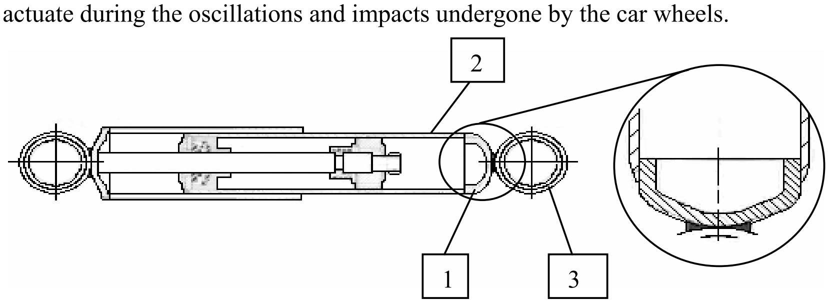

Figure 1. An automotive shock absorber and a detail of the evaluated part.

The end cap is manufactured through sheet drawing, involving the pressing of a punch over a circular blank, which is located over a cylindrical die, as illustrated in Fig (2). The figure also shows the dimensions in mm, as well as the blank holder, on which a constant load is applied. The feeding of blanks to the press is performed manually.

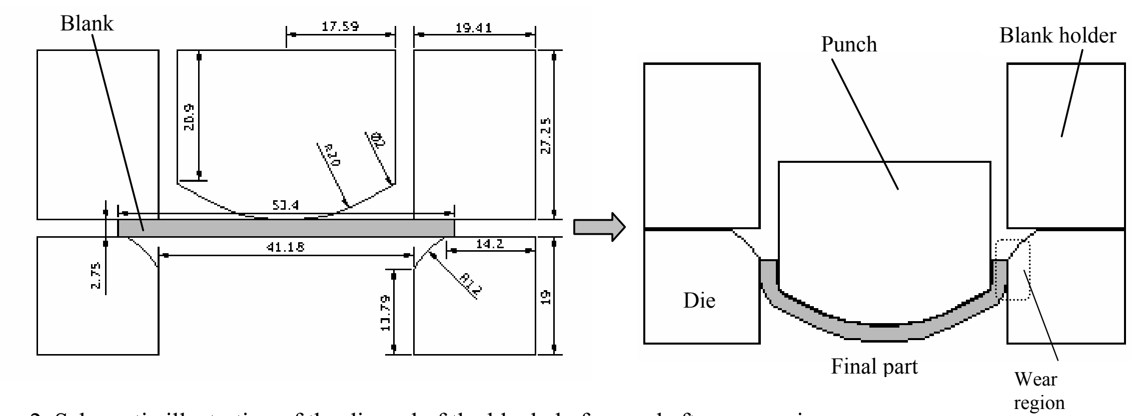

Figure 2. Schematic illustration of the die and of the blank, before and after processing.

The blank material is an AISI 1008 carbon steel, 26,7mm in radius and 2,75mm thick. The die is made of a standard tool steel, followed by superficial polishing in the die/material contact surfaces. Die life is controlled by the possibility of completing successive die polishing without reaching the demanded dimensional characteristics. Die fracture is obviously intolerable for the manufacturing process.

2. Premature Die Wear

A premature wear of the drawing die throat was observed, in the region indicated in Fig (2). This resulted in an unacceptable scratching of the external surface of the produced endcaps. Such damage led to frequent production interruptions, in order to re-polish or replace the die, which also occasionally underwent fractures.

Figure (3) displays the external lateral aspect of a scratched endcap, observed through optical stereomicroscopy (larger photo on the left) and through Scanning Electron Microscopy (SEM) (other photos). The damage, originated from the contact with the damaged die wall, can be clearly seen. SEM was performed under a 15 kV voltage and the samples underwent an initial ultrasonic cleaning in acetone, in order to remove any organic residue in the endcap. The faulty surface finish shown in the picture demanded an additional tumbling operation, in order to smoothen their lateral walls and making possible the subsequent manufacturing steps of the shock absorber. The aspect of the damage is typical of adhesive wear (Wulpi,1985, Dieter, 1997), and is usually associated with excessive pressure on the die for the prevailing lubrication conditions (Ashby, 1992).

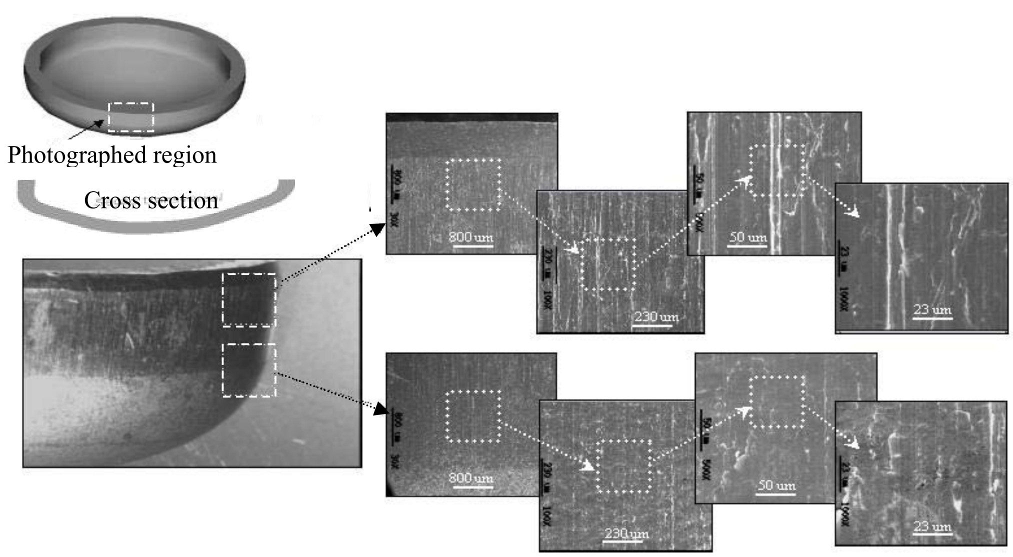

Figure 3. Details of the scratched regions in the shock absorber endcap.

As will be soon shown, the drawing of the endcap involves a thickening of the blank in its final stages. This introduces high compressive stresses as the material is processed between the die and the punch, causing the superficial damage in the die throat.

3. Finite Element Simulation

The formulation of the finite element analysis in the plastic range utilizes the concept of discretization and of variational principles, similarly to the analyses in the elastic range (Kobayashi, 1960, Zienkiewicz, 1969). The domain is initially divided in discrete elements, and a form function is used for each element. This function is defined for a local coordinate system in the element, allowing the calculation of the velocities for all points in the element, in relation to a general coordinate system.

Once the velocity field is obtained as described in the previous paragraph, the velocities are introduced in a variational representing the material to be simulated. Equation 1 describes such a situation for a rigid-plastic material. The symbols in the volume integral correspond to the von Mises effective stress and strain rate, and in the area integral they correspond to forces and velocities.

$$\pi = \int_V \bar{\sigma}\,\dot{\bar{\varepsilon}}\,dV - \int_{S_F} F_i u_i\,dS \tag{1}$$

The minimization of the variational generates a set of non-linear equations, that is solved using matrix approaches and numerical methods such as the Newton-Raphson one. The solutions for these equations supply the velocity values for all points of the material, which will in turn allow the calculation of strains, strain rates, stresses and loads on the part being formed or on the dies (Kobayashi, 1973, Zienkiewicz, 1974).

The forming process under discussion was simulated utilizing the finite element method, employing the commercial software DEFORM® 7.0 for Windows (Scientific Forming Technologies Corporation, Ohio, USA). The geometry was initially the one already described in Fig (2). Two other geometries of the punch were also considered, aiming at a reduction of the loads in the die. Figure (4) illustrates the four parts of the drawing dies, which were taken as rigid (no elastic deformation) in the numerical simulation.

The process was considered as axissymmetric, which greatly simplifies the numerical analysis of the problem and leads to a large economy in processing time in the computer. Several meshes were analyzed for the blank, in order to speed the computing time and increase the precision of the results. An insufficient number of elements in the mesh may cause numerical instabilities. These instabilities lead to pronounced oscillations in the calculated loads on the dies, during the simulation.

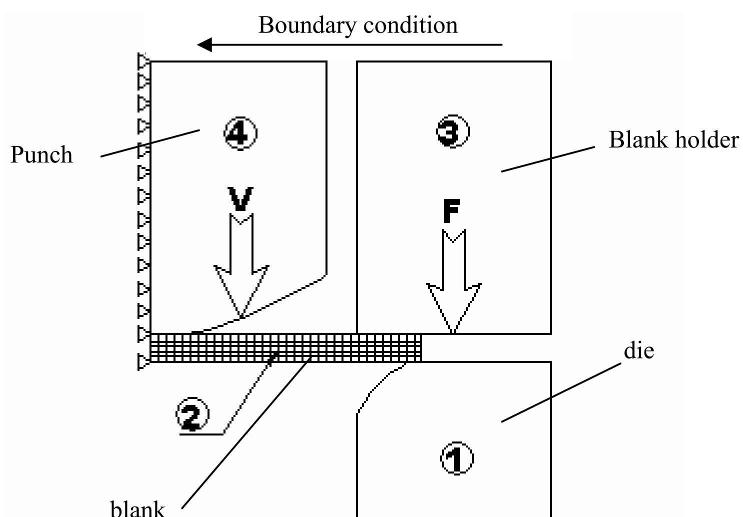

Figure 4. Illustration of the dies used in the DEFORM program, including the employed bounday conditions.

A mesh of 3000 elements was employed for the blank, causing load oscillations on the tools of about 8% of the maximum observed loads. The computer time for such a mesh was about 5 times that for a mesh with 500 elements,, which, on the other hand caused load oscillations of up to 45% of the maximum observed loads. The mesh included regions with different densities of elements, allowing the use of a higher number of elements in the regions subject to higher strains, close to the outer borders of the blank. The element density in this region was 5 times that in the central region of the blank.

The die did not move during the simulation. The punch moved at a speed V = 10 mm/s, and the load on the blank holder was 1,0kN. The stress-strain curve for the material (AISI 1008) was the one in the internal library of DEFORM®. The time for the movement of the punch in each step of the simulation was 0,10s, and the simulation was run for 225 steps. Three simulations were completed, as described in Table 1. The first one followed the original drawings, the second considered a necked punch, and the third one a conical punch (see Fig. (5)).

Table 1. Numerical Simulations

Name	Characteristics
CA-0-1	Geometry identical to the original one
CA-0-2	Necked punch
CA-0-3	Conical punch

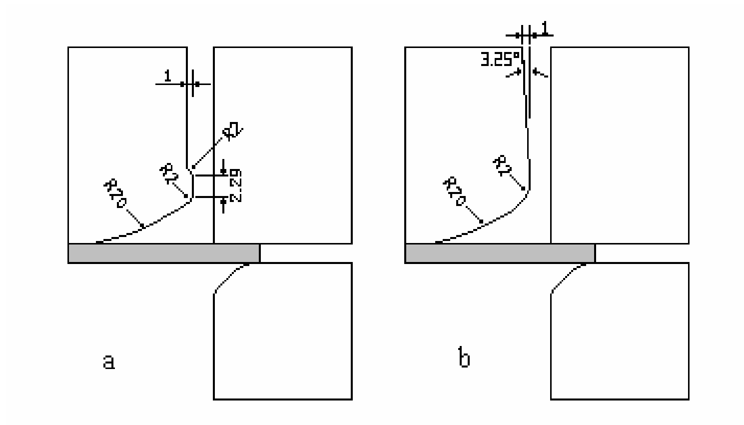

Figure 5. Illustration of the simulations CA-0-2 (a) and CA-0-3 (b)

4. Results and Discussion

Figure (6) displays the finite element meshes of the endcap after the simulation, for the three considered cases.

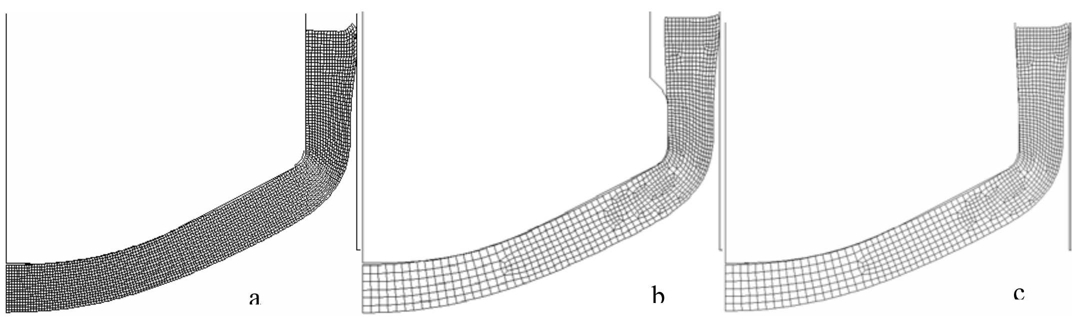

Figure 6. Finite element meshes at step number 225, for simulations CA-0-1 (a), CA-0-2 (b) e CA-0-3 (c)

Figure (7) shows the variation of the die loading with time, along the simulation of the process. The radial load in the die (X direction) can be quite high, reaching almost 100 kN. This is due to the observed thickening of the outer parts of the blank, which must pass between the die opening and the punch in the final stages of the process, as illustrated in Fig. (8). It is believed that this excessive loading is the dominant aspect causing the premature wear in the die throat, which then causes the superficial damage in the outer walls of the endcaps, or the breakage of the die.

The radial load can be decreased by an increase in the clearance between the die and the punch. The die opening, however, cannot be changed, since the endcap must fit into the shock tubular body before the welding step. The geometrical changes in the punch utilized in simulations CA-0-2 e CA-0-3 do not cause changes in the external dimensions of the endcap, but allow a decrease in the radial loads in the die.

The changes in punch geometry under consideration do not alter either the cup part of the endcap or the external diameter. Figures (9) and (10) display the loading on the die in directions X and Y, according to the simulations performed with DEFORM®. The loads in the X direction were reduced: for simulation CA-0-1 its maximum value was 97,6kN, which decreased to 84,2kN in simulation CA-0-2 (13,7% lower), and to 84,1 kN in simulation CA-0-3 (13,8% lower). On the other hand, loads in the Y direction are not affected by the changes in punch geometry.

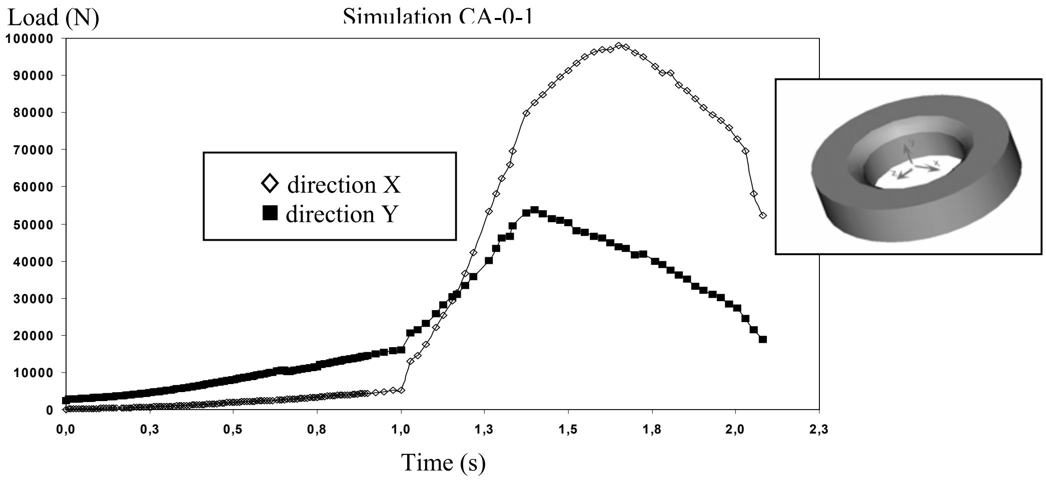

Figure 7. Loads on the die for the simulation CA-0-1

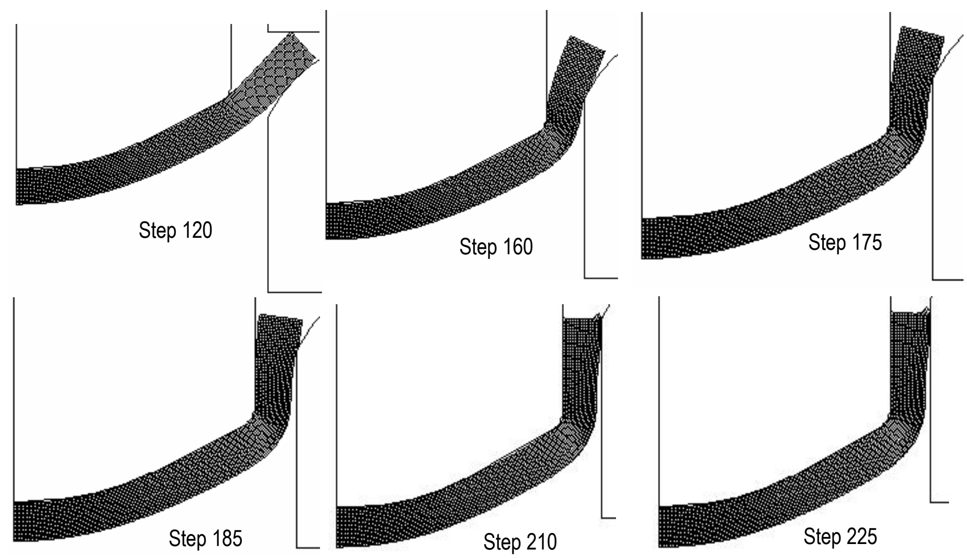

Figure 8. Changes in blank thickness caused by the simulation CA-0-1

4.1. Dimensional Comparison between the Real Part and the Predictions of the Simulations

A dimensional comparison between the real part and the numerical predictions was performed considering the measurement of the endcap thickness in 12 points along its length. Figure (11) illustrates the position of these points.

The endcap thickness predicted numerically was obtained using the measuring tool included in DEFORM, which supplies the distance between two specified points in the part. The thickness in the real part was measured after a longitudinal sectioning of an endcap, which allowed the measurement of its thickness in a MITUTOYO PJ311 shadowgraph, with a magnification of 20X and utilizing a digital measuring system (precision of 0,001 mm). Figure (12) displays the thickness results for the real part and for the simulations.

For the case of the simulation CA-0-1 (corresponding to a cylindrical punch, as used presently) the differences between the thickness in the real part and the simulation ranged from a minimum of 0,83% to a maximum of 3,92%. The same numbers for simulation CA-02 were -5,9% and 2,18%, and -3,9% and 4,53% for simulation CA-0-3.

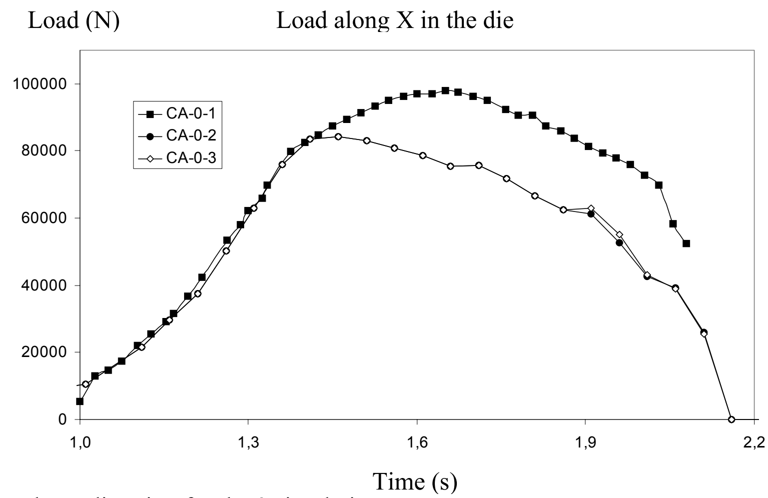

Figure 9. Die loads along the X direction for the 3 simulations.

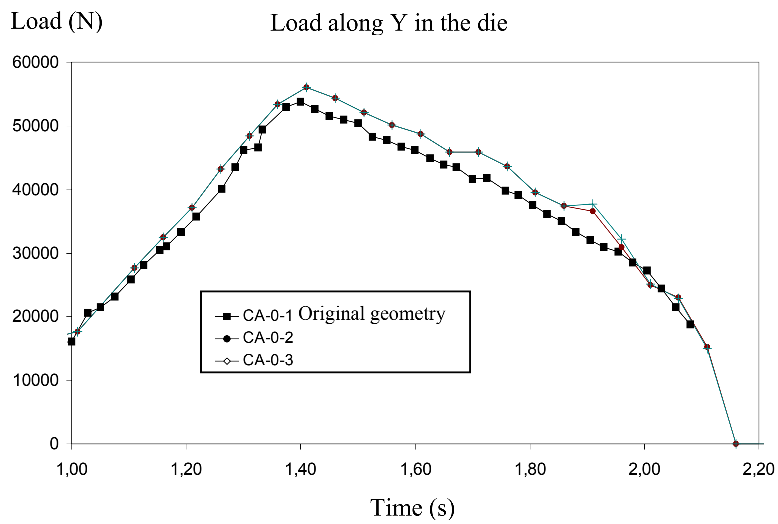

Figure 10. Die loads along the Y direction for the 3 simulations.

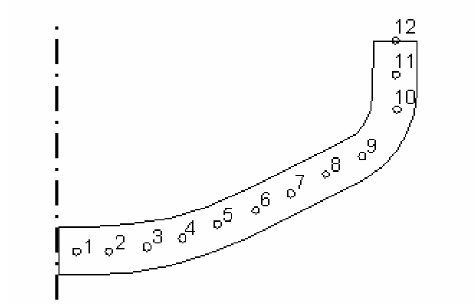

Figure 11. Points along the endcap were the thickness was measured for the real part and for the simulations.

5. Conclusion

The numerical simulation of the drawing of a shock absorber endcap, utilizing the software DEFORM®, supplied a good dimensional similarity with the real part. This leads to a reasonable validation of the present numerical simulation of the forming processing of the part under consideration.

The simulations indicated that as the material in the blank is deawn into the die, there is an increasing thickening of the blank, which must then be ironed between the punch and the die throat. This ironing leads a high level of loading on the die during the final stages of processing.

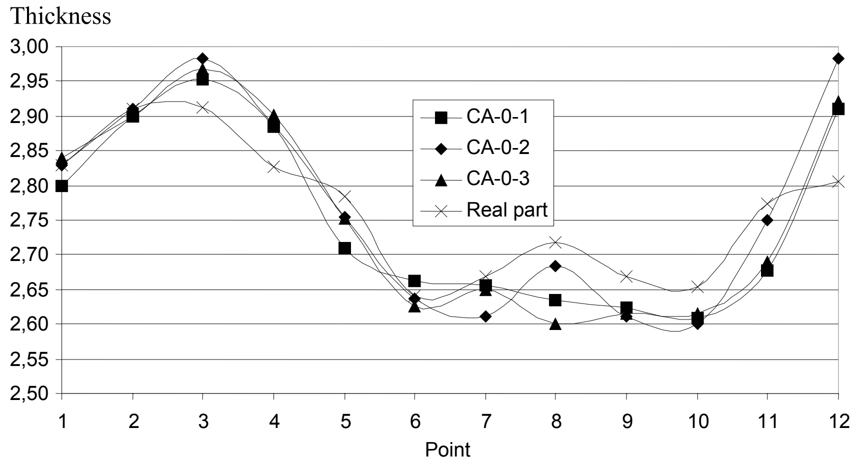

Figure 12. A comparison between the thickness of the real part and of the simulated endcaps.

The observed premature wear damage in the die is caused by the overloading caused by the ironing. A numerical analysis of different shapes for the punch lead to decreased die loading, without dimensional problems for the endcap, and represent a solution for the processing failure.

6. Acknowledgements

The authors gratefully acknowledge the financial support of CNPq (Conselho Nacional de Desenvolvimento Científico e Tecnológico), and of PRONEX (Programa Núcleos de Excelência - CNPq and FINEP).

7. References

Wulpi, D. J., Understanding How Components Fail, Chapter 11, p. 163, 1st. ed., American Society for Metals, Metals Park, Ohio, USA, 1985.

Dieter, G. E. (Ed), Metals Handbook, vol. 20 (Materials Selection and Design), Design for Wear Resistance, p. 603, ASM International, Materials Park, Ohio, USA, 1997.

Ashby, M. F., Materials Selection in Mechanical Design, 1st. Edition, Pergamon Press, Oxford, 1992.

Kobayashi S., McDonald V., Thomsen E. G., Some Aspects of Press Forging, Int. Journal Mechanical Science, 22, p. 91-98, 1960

Zienkiewicz O. C., Valliappan S. King I. P., Elasto Plastic Solutions of Engineering Problems "Initial Stress " Finite Element Approach, Int. Jour. Num. Meth. Eng., 1, p.75-100, 1969

Kobayashi S., Lee C. H., Deformation Mechanics and Workability in Upsetting solid Circular Cylinders, Proc. North American Metalworking Research Conference, NAMRC, 1, 185, 1973

Zienkiewicz O. C., Godbole P. N., Flow of Plastic and Viscoplastic Solids with special Reference to Extrusion and Forging Process, Int. Jour. Num. Meth. Eng., 8, p.3, 1974