DIE PROJECT AND FEM ANALISYS FOR AUTOMOTIVE COMPONENT PRODUCTION

Lora, F. L., lora@cimatec.fieb.org.br [1]
Yurgel, C. C., chemale@cimatec.fieb.org.br [1]
[1]SENAI CIMATEC, Avenida Orlando Gomes, 1845 – Piatã – CEP 41650-010 – Salvador – BA – BR

Folle, L., luis.folle@ufrgs.br [2]
Schaeffer, L., schaefer@ufrgs.br [2]

[2]UFRGS-LdTM, Av. Bento Gonçalves, 9500 – Agornomia, CEP 91501-970 – Porto Alegre – RS – BR

Abstract. *Nowadays many pieces used in the automotive industry are produced by metal forming processes. In the case of sheet metal forming the processes are cut, bend and metal spinning; in cars manufacturing industries, bodywork, intern pieces. The objective is to demonstrate the development of a tool project for support to brake cable, with computer validation of the process of tjis paper. Before its fabrication, it avoids the trying and error method. This computer validation of the tool using Dynaform software, indicated the zone with major mechanical stress (Strain and Stress), the thickness of the piece and the geometry of the preformed in each step of the process. This work was developed in the extent of a innovation program fomented by National Department of SENAI.*

Keywords*: support to brake cable, tool project, computer validation.*

1. INTRODUCTION

Recently the State of Bahia has been experiencing the successful installation of companies and of final product for the consumer, such as automotive line (FORD, Sodecia, BSB, Ferrolene, Arvin, Krupp, Forja Bahia, Centrotampa, Gerdau and Fixar), as well as white line (Britânia, Faet and Mondial). The installation of these companies has also brought the necessity for a number of local suppliers, components as well as systems in order to decrease the cost of products by reducing levels of logistics and taxes, as well as technical assistance.

Currently, most components used in the automotive industry are produced by the process of sheet metal forming, which includes the processes of cutting, bending and stretching, taking as examples the body of the car, internal components of the panel, structural components, a great attention has been paid worldwide to reduce the volume and weight of products, Bor-Tsuen Lin 2009. Sheet metal forming process is an important technology in manufacturing, especially in the automotive industry, M. Samuel 2004. An important concern in whether the desired deformation can be accomplished without failure of the work metal, M.H. Chen 2007.

The final goal of the process of manufacturing is a product from a blank or from sheet strip, which will provide the desired shape and dimensions, without defects or failures. The format of the final product after processing is defined by tooling, blank and parameters of the process. Errors in the design of the tool or the format of the blank, and the inappropriate choice of material to be processed and the parameters of the process, can generate a product with inadequate geometry or failures.

Inputs required to simulate a sheet metal-forming process are computer-aided design (CAD) model of the forming tools (die, punch and binder), mechanical properties of the material used and other process parameters like binder load, lubrication, geometry, etc, according to K. Hariharan 2009.

The objective of this study is to show the methodology to manufacturing an automotive component tool for sheet metal forming, so that the condition is working with high production and reduced costs.

2. METHODOLOGY

In developing tools for the manufacturing process of sheet metal forming the time to design and tryout later becomes the parameter of great influence in the cost of industries. In the tryout, step that is made to validate the tool in equipments, errors like burrs and springback may appear in the piece that were not anticipated in the tools design stage, so the tools need a rework generating additional time and consequently higher cost.

This work made use of the reverse engineering methodology for the production of a component used in the automotive industry to develop a tool for producing it. Part of the design was planning the piece, pre designing of the tools, and validation the tool via FEM and fabrication of the tool.

2.1. Component Geometry

The product to be produced by the tool is the support to brake cables used in passenger cars, which function is to support the brake cables that pass underneath the vehicle. This piece is surface treated to increase resistance to corrosion in the environment which it operates. The design of the component to be manufactured is shown in Figure 1 and its thickness is 2mm during all piece.

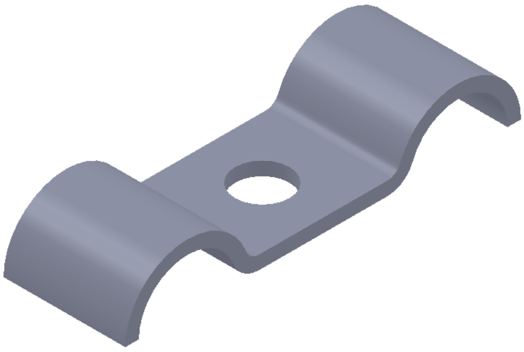


Figure 1. Design of the workpiece.

2.2. Tooling planning

In the development of the tool a pre-project is carried out to estimate the stages of component manufacture. The first step in the development of tools to forming the achievement of the pre-project to design the dies and punches that will give the final shape to the component.

Some definitions should be taken in this stage of development, such as:

a) Process to be carried out in a progressive tool (several steps within the same tool), requiring only one tool for the component production;
b) The manufacture of the component will be necessary the cutting and bending steps;
c) The maximum possibility of using the sheet metal in the cut end bedding operation, according to Eduardo 1997;
d) Conventional Cutting Technique: production of pieces with medium values of tolerance, with medium frequency of the machine. This process is done in mechanical presses type C structure, Hellwig 1996;
e) Use feeding strip tool;
f) The strip feeding process is held by a feeder in the tool, so the process will become automatic. With this type of feeding the strip needs to be provided by a coil;
g) Placing of strip guide pins to help the alignment of the strip in the tool;
h) Increase of the length of cutting tools used to separate the pieces, making the cut in the end of progressive tooling thus ensuring the exclusion of cut marks on the sides of the piece;
i) After cutting step the strip is no longer guided by the strip guide pins, however it is necessary that the strip remains guided. The holes pilots are in the central of the strip with the last step performing the hole calibration as part of the work piece design;
j) The bending punches should be aided by springs to ensure the smooth operation even with a possible lack of displacement of the hammer of the press;
k) The tool must have a intermediate lifter guided by pins to assure the alignment between the punch and the die;
l) Use of a blank holder through the guide pins fixed by compression springs.
m) In the cutting step, the clearance between punch and die is referent the sheet thickness (die clearance is approximate five hundredth of the thickness), Provenza 1993;
n) At shearing stage, the clearance between punch and die is equivalent to the sheet thickness.
o) At bending stage, the spring back effect must be considered and estimated previously.

With the pre project definitions at hand, a piece planning was achieved. To estimate the blank dimensions before conformation; the work piece was sketched in CAD software with its final shape and the length of the middle line of the cross section was measured to obtain the width of the blank. This is possible because the work piece has bending only in the longitudinal direction.

After tooling pre-project, it is possible to validate it by means of numerical simulation, avoiding most project faults and reducing the try-out time. Picture 2 shows the tooling planning diagram and the progressive step operations.

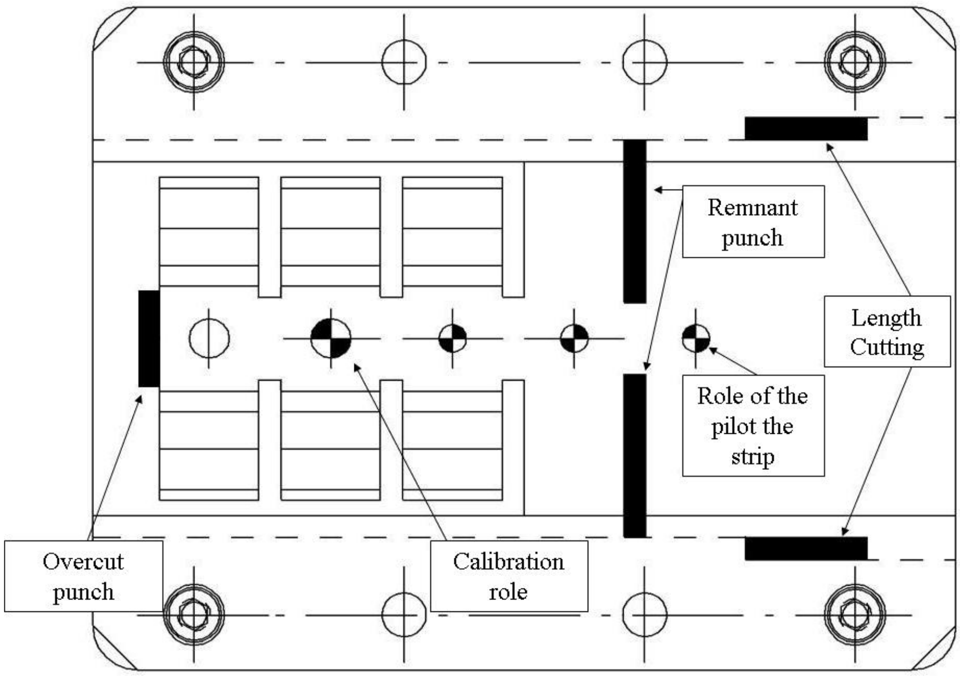


Figure 2. Tooling planning diagram showing the progressive step operations.

2.3. Computational Simulation

The computational simulation is used to validate tooling based on the parameters of the planning phase like: materials proprieties, tool geometry and process variables (Punch speed, strength, displacement, among others). The simulation software used was eta/Dynaform, a finite element simulation and solver program focused on sheet metal forming.

The material proposed for manufacturing is AISI 1008 steel; its proprieties were taken from the software library and described in table 1.

Table 1. Workpiece materials proprieties.

Proprieties	Value
Material	ABNT 1008
Young's Modulus	207GPa
Poisson's Coefficient	0,28
C (Strength coefficient)	520MPa
n (work hardening coefficient)	0,24
R0 (anisotropy at 0º)	1,86
R45 (anisotropy at 45º)	1,72
R90 (anisotropy a 90º)	2,28
Ultimate Stress	441MPa
Yield Stress	126MPa
Ultimate Strain	0,5

In the simulation process many parameters were specific in the agreement of the real process and these parameters are described in table 2.

Table 2. Parameters of the simulation process.

Parameters	Value
Speed Punch	15mm/s
Friction Coefficient	0,125
Yield Criteria	Hill 1948
Blank Geometric	76 x 18 x 2 mm

Some particular properties of the ETA/ Dynaform are described below:

- Contact interface: "Form One Way Surface to Surface": This kind of contact is used to model rigid bodies that use shell type elements to which it is necessary that the thickness of the rigid body is taken into account;
- Process Type: "Single Action": The movement of the tool is related to the type of displacement of the same toll (Draw Type). Defined as a Single Action, the movement is performed by dies and punch and remains static, and both the displacement as the forces of blank holder are assigned to the dies;
- Material Property: Belytschko-SBL. The Property Belytschko-Tsay is used to define the type of mathematical formulation that will be applied on elements of type shell workpiece. This property is widely used in simulations of stamping.

The element used to define the tools (punch, die and press plates) is also the type shell, but these elements are undeformable.

2.4. Tooling Project and Fabrication

The tooling project will be concluded when all components are discriminated, like springs, lifter, screw. After tooling project is concluded, it is sent to fabrication through processes of machining and thermal treatment.

3. RESULTS ANALISYS

3.1. FEM Analysis

The FEM analysis examined the manufacturing process and the response of the material to the stress and strain applied. The FEM software used was DynaForm 5.6 that is dedicated to sheet metal forming analysis.

The figure 3 shows the result of the FEM analysis for the first step of the manufacturing process which is the bending part. The strains of the material are all in the safe part of the FLC (Forming Limit Curve, in the left of figure 3). This result shows that manufacturing problems like cracks and wrinkling in the material probably will not occur. However in the ends of the workpiece an unwanted bend will appear because the final shape of the workpiece does not contain these bends. This defect needs to be avoided in the first step of the manufacturing process because the next steps are the sizing of the final shaped form in order to prevent an extra step to unbending the ends of workpiece.

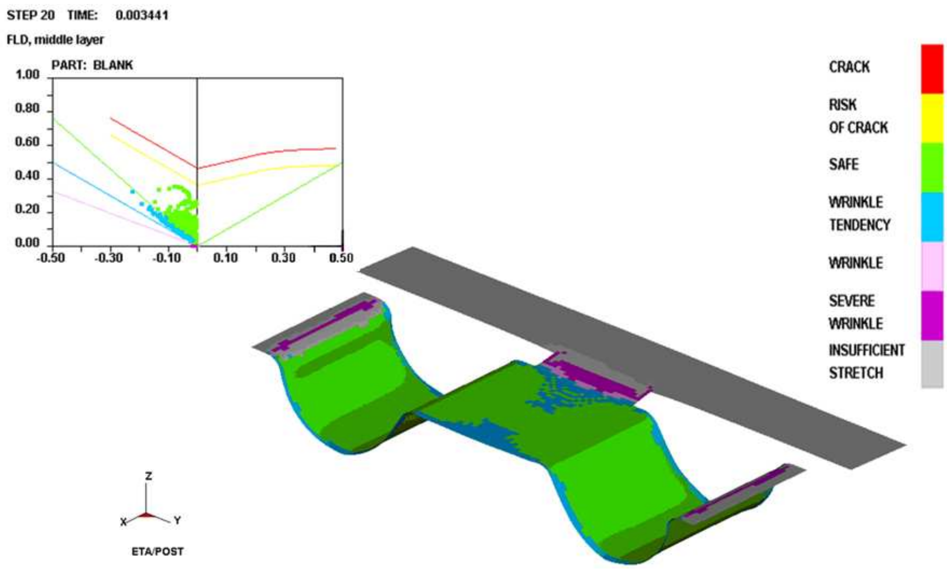


Figure 3. Worckpiece forming with flaps defect.

In order to avoid the bend of the workpiece ends, in the first step of the bending, was made a rounded chamfer in the die to improve the flow of the material in that place. The figure 4 shows the FEM analysis after the die modification. It can be seen that the material rarely exhibits the bending in the ends of the workpiece.

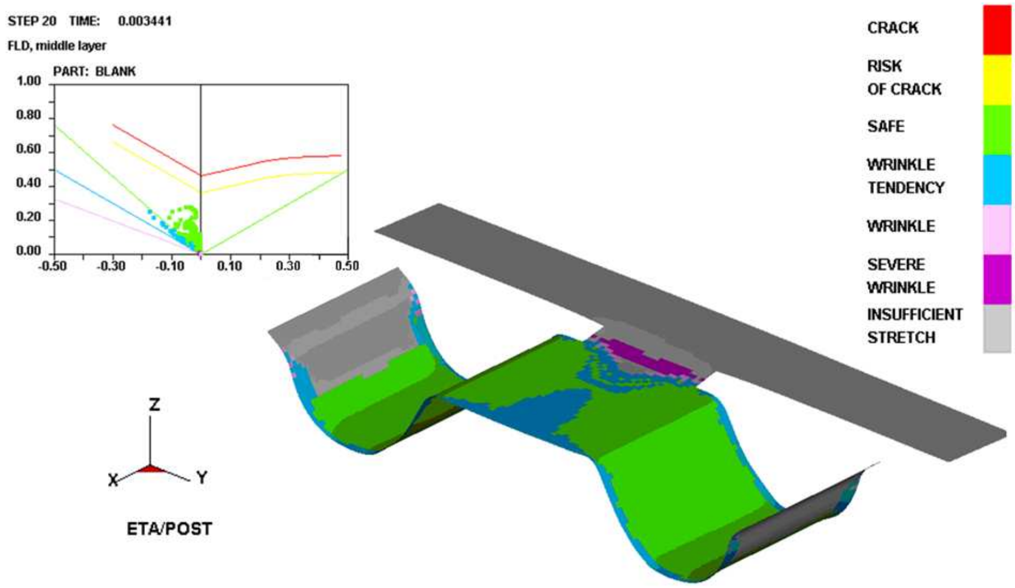


Figure 4. Worpiece without the flaps with bending.

At figures 3 and 4, it can be noted that the workpiece is joined to a strip with the same dimension of the blank. This strip was placed purposely in the analysis because, during the first step in the real manufacturing process, the workpiece was still joined to the sheet strip that led it. Since the analysis has the function of preventing what will happen with the work pieces, these have to be performed with the maximum fidelity as the real case and it explains the strip placement that is joined with the blank.

3.2. Tooling Project and Fabrication

After the end of simulation, to validate the preplanned tool, its definite project is done, with the project accomplished, the next step is sent to tool shop for the tooling manufacturing. In the figure 5, it can be noticed the tooling die; in detail as well as the dimensioning of all components that will be necessary to its functioning shown the necessary chamfer for the component forming, as indicated by FEM analysis.

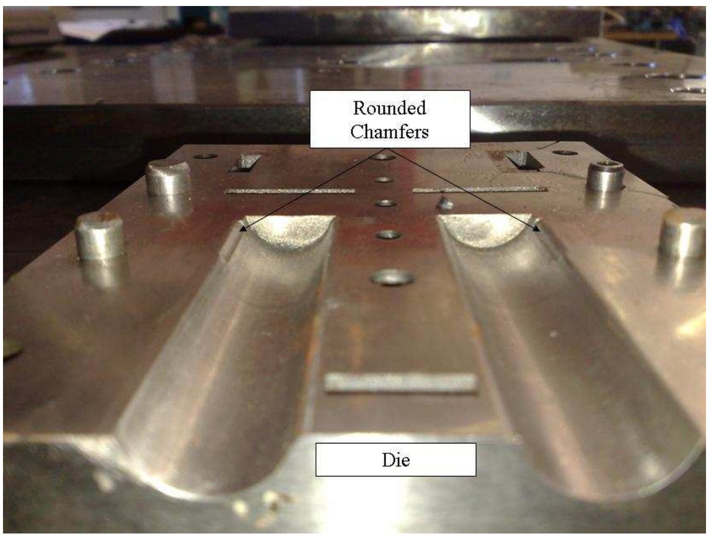
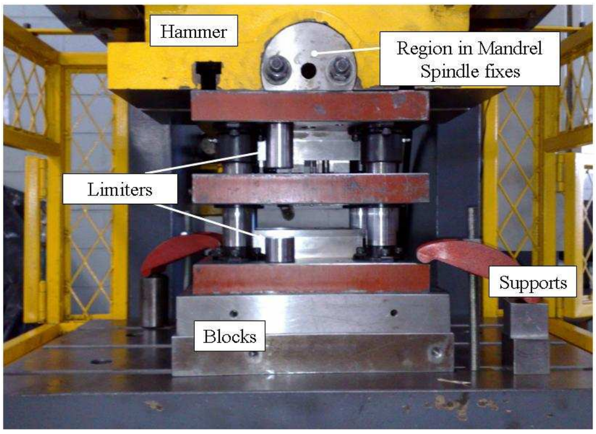


Figure 5. Detail of chamfer generated in the die.

To perform the tooling tryout it was used an 40ton eccentric press, in which can be observed in the figure 6 the tooling inserted in the press and manufactured components. In figure 7, it is noted the produced component by the manufactured tooling in the project.



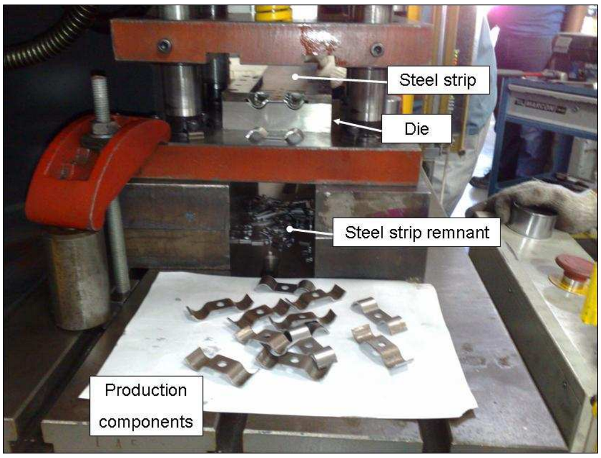


Figure 6. Positioning of stamp in the press and manufactured components.

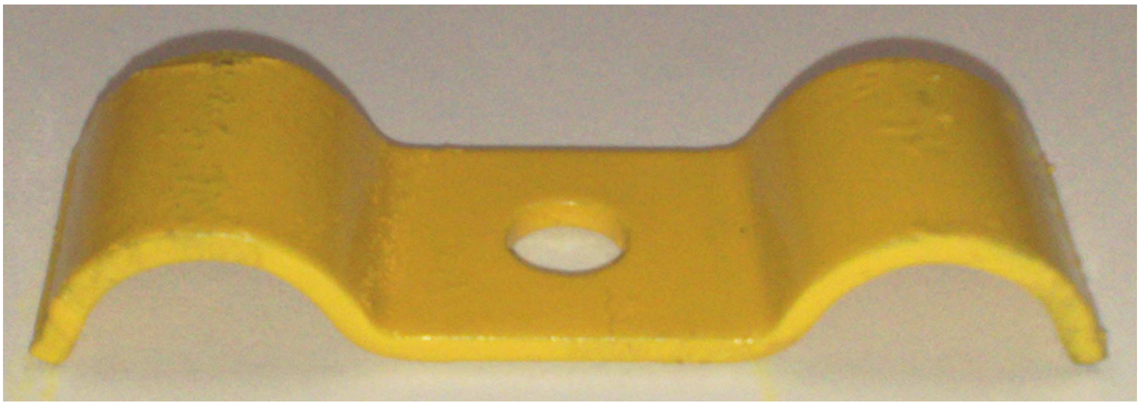


Figure 7. Piece produced by the manufactured tooling in the project.

4. CONCLUSIONS

- The methodologies to develop projects must be followed to avoid possible failures and elevated costs to manufacturing pieces;
- The achievement pre-project should be discussed among the team members to define steps and parameters to be used; it helps to decrease the project development time;
- The numerical simulation was fundamental for the project to decrease the tryout time, whereas a possible failure in the tooling was found before the tooling project performance;

It can be deduced with this project that the numerical simulation help the development of sheet metal forming tools, cutting off costs and possible reworks that will occur after the tooling tryout.

5. ACKNOWLEDGEMENTS

The authors acknowledge the promotion and the help during the development provided by SENAI DN. The authors are also grateful for the cooperation with the CARHEJ Nordeste Company.

6. REFERENCES

Bor-Tsuen Lin, Chun-Chih Kuo. APPLICATION OF AN INTEGRATED RE/RP/CAD/CAE/CAM SYSTEM FOR MAGNESIUM ALLOY SHELL OF MOBILE PHONE. Journal of Materials Processing Technology. Vol 209 (2009) Pg. 2818–2830.

Eduardo Luiz Álvares Mesquita, Léo Lucas Rugani. ESTAMPAGEM DOS AÇOS INOXIDÁVEIS. DEZEMBRO – 1997.

Hellwig, W., Semlinger, E. SPANLOSE FERTIGUNG STANZEN, Vieweg Verlag, Wiesbaden, 1996.

K. Hariharan, C. Balaji. MATERIAL OPTIMIZATION: A CASE STUDY USING SHEET METAL-FORMING ANALYSIS. Journal of Materials Processing Technology. Vol. 209 (2009) Pg. 324–331

M. Samuel, Numerical and experimental investigations of forming limit diagrams in metal sheets, J. Mater. Process. Technol. Vol. 153–154 (2004) Pg.424–431.

M.H. Chen, L. Gao, D.W. Zuo, M. Wang. APPLICATION OF THE FORMING LIMIT STRESS DIAGRAM TO FORMING LIMIT PREDICTION FOR THE MULTI-STEP FORMING OF AUTO PANELS. Journal of Materials Processing Technology Vol. 187–188 (2007) Pg. 173–177.

Provenza, F. ESTAMPOS II, Editora F. Provenza, 1993.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.