STUDY OF THE PROGRESSIVE SHEET METAL FORMING PROCEDURE OF AN INTERCOOLER PART

Victor Alexandre Ramos Castilho¹a
¹Instituto Superior Técnico,
Avenida Rovisco Pais, 1049-001 Lisboa, Portugal.
a_victor.castilho@tecnico.ulisboa.pt

Mentors: Prof. Bárbara Perry Gouveia, Prof. Jorge Rodrigues
May 2017

Abstract
The present work has the objective to study four steps of the metal forming procedure of steel plate used in automotive intercooler, from the aluminum alloy AA4343 (7.5%) – HOGAL 3551 – 7730 (7.5%) with 1.6 mm of thickness. This part is already produced by a company called JDEUS and during the start of the production it has appeared some fractured stamps during a stamping process. This piece is done with progressive sheet metal forming tools in fourteen steps.

In this work it’s used a finite element program to simulate the four forming steps part of the procedure to achieve the final geometry. The objective is to know the required punch strength in each forming step and finding the reasons and a possible solution for the existing problem. In order to do that it involved the testing of the material in the laboratory and the later matching of the experimental data with the data of numeric model of the same specimen of test done in LS-DYNA. After the definition of the anisotropic material model with success, the simulations were done for the four steps, importing the deformed shape of the sheet from first step to the next.

The results from the simulations allowed to understand the punch force evolution during each forming step, including the mono block effect, thickness reductions and a solution suggestion is given to avoid defects that appeared in the first usages of progressive tool.

Keywords: Progressive sheet metal forming, Anisotropy, Finite element method, LS-DYNA

1. Introduction

At present, sheet metal forming processes are widely used in various industrial sectors such as automotive, electric home appliance, and aircraft industries. Also, the needs for high precision and high value-added products are increased. In order to reduce the process development time, the prediction of defects and the modification of design in the design stage are needed.

Virtual prototyping tools of processes and systems in the domain of sheet metal working is nowadays a real prospect for industrial users to provide accurate predictions of the part geometrical features and post-forming characteristics (e.g. residual stresses) and possible defects and failures on the basis of the chosen process parameters. Thanks to these predictions, critical decisions in process design are taken, strongly affecting the technical and economic success of the process.

In sheet metals, the response to plastic deformation manifests itself through different phenomena, such as hardening, anisotropy,
failure and fracture. Most of these phenomena occur simultaneously with significant interactions and may deeply affect the behaviour of the sheet metal due to the significant changes they cause in its physical and mechanical features and properties, such as surface appearance and roughness, yield-point elongation, resistance to plastic deformation, hardness and strength, residual stresses and geometric distortion, springback, and formability.

The Forming Limit Diagram (FLD) is the most popular criterion for predicting failure in sheet forming operations. It indicates the combination of the major and minor strains (ε1 - ε2) that can be applied to a metal sheet without failure. The strain domain covered by the FLD must replicate as close as possible the various strain states arising during industrial sheet forming operations: to achieve this, various strain paths are applied to the metal sheet, ranging from equibiaxial tension (ε1 = ε2) to pure shear (ε1 = −2ε2).

The approach based on FLDs to predict the sheet failure is still the most widespread, especially in industry, and is currently implemented in the finite element commercial codes devoted to sheet forming simulation, mainly thanks to its simplicity in application. However, it presents several drawbacks, which have pushed the development of alternative approaches for predicting the sheet failure. The FLDs depend on the sample thickness, fall short in the case of non-linear (or non-proportional) strain paths, which very often characterize the sheet forming processes.

2. Overview

A metal forming process is a set of mechanical procedures by which a strip or a blank is deformed into an object of specific shape, whether plane or hollow, in one or more steps. Because of the plastic deformation the sheet is subjected to, the acquired configuration is permanently maintained. In particular, the drawing of metal or deep drawing is the process in which a flat sheet of metal is formed into a cylindrical-, conic- or box-shaped part. The final workpiece has to be achieved using minimal operations and generating minimal scrap, meeting anyway definite quality requirements [1].

The typical machinery involved in an usual sheet metal forming procedure is constituted by the following tools:

- **Die**: it represents the base on which the blank is initially placed. The die is characterized by a cavity whose contour reproduces the profile of the final work piece. Its depth is related to the height the drawn piece has to achieve;
- **Punch**: it is the tool by which the blank is forced to flow into the die cavity. The punch is designed in a way that allows to obtain the expected form at each stage of the process. The face of the depends on the forming process and on the final mold.
- **Binder/Blank holder**: it is usually positioned over the blank with the aim of slowing down its flux into the die. The force applied by the binder is called “Holding force”, thus distinguishing it from the “Drawing force”, exercised by the punch. This tool is notably useful in order to avoid blemishes on the work piece borders.

3. Case of study

The intercooler part, the head plate, named “Chapa testa Giorgio 2SV”, its initial forming steps are analyzed in this work and it can be seen in Picture 1. The production process of this plate involves a progressive stamping machine with 14 steps include sequentially: 3 cutting stations; 4 forming stations (in the present study); 2 exterior cutting stations, 1 side bending station, 1 station for tear opening, 1 another bending, 1 unwarping station and a final cutting station. The material is automatically feed with plate stripe of aluminum AA4343 (7.5%) – HOGAL 3551 – 7730 (7.5%) with 1.6 mm thickness and 318 mm width.
The study of the four forming steps is done with commercial dynamic finite element solver LS-DYNA® [2]. The objective is to estimate the evolution of punch force used during each step, understand the effect of the mono block in the closing of the progressive tool, thickness changes, and the forming limits of the actual production that could affect the quality of the final product.

The head plate is a plate with high detail that separates the rest of the intercooler from the exit box, as shown in Picture 2.

Progressive tools are frequent in sheet metal forming industries. They execute multiple operations in cycles separated from superior neutral point (SNP), to the inferior neutral point (INP), related with press functioning that is mounted in the progressive tool.

The design of the plate stripe with respective sequence of stations is available in the annex A.

In the start of the production of the head plate, some samples shown a defect appearing in the second forming step, as show in Picture 3.

3.1. Numerical and experimental determination of the main process variables needed for numerical simulations of the forming steps.

To properly simulate the forming steps, it requires to know a set of process variables to reproduce reality with a numeric simulation. e.g. Mechanical characteristics of the raw material, Binder force, friction coefficient in the interfaces of the tools. To start this work the first step was to properly characterize the mechanic behavior of the material.

3.1.1. Mechanic characterization of the aluminum alloy AA4343 (7.5%) – HOGAL 3551 – 7730 (7.5%)

The uniaxial tensile test according to the ASTM E8 standard [16] is still nowadays the most widely used testing method for determining the sheet metal behavior, mainly thanks to its intrinsic simplicity of execution. By using specimens machined at 0°, 45° and 90° with respect to the rolling direction, has shown in Picture 4, the yield stress and the anisotropy coefficients along the three directions can be evaluated, providing the basis for the calibration of most of the anisotropic yield criteria and hardening models.
The results from the uniaxial tensile tests allowed to see the effects of the anisotropy, in Picture 5 the true strain and stress are plotted for the 6 specimens analyzed.

This mechanical behavior can be described by the Ludwik-Hollomon [3] empirical equation.

\[ \sigma = K \varepsilon^n \quad (3.1) \]

Where K is the strength coefficient, and n the hardening exponent, this parameters can be determined by the linear approximation of the true strain-stress logarithmic curves. To do that its used the following formulation,

\[ \sigma = Ke^n \iff \ln \sigma = \ln K + n \ln \varepsilon \implies (3.2) \]

The approximation is the characteristic equation of a line like,

\[ y = mx + b \quad (3.3) \]

With this, m and b values can be taken from the respective ln(\sigma) vs ln(\varepsilon) plots, and combining the equations 3.2 and 3.3, we see that m=n and b=ln(K).

The young modulus, E, yield stress, Se, rupture Stress, Sr, Poisson coefficient, \( \nu \),

\[ \nu = -\frac{\varepsilon_w}{\varepsilon_l} \quad (3.4) \]

were also calculated, the anisotropy coefficient, \( r \), are calculated with true strain like:

\[ r = \frac{\varepsilon_w}{\varepsilon_n} = \frac{\varepsilon_w}{-(\varepsilon_w + \varepsilon_l)} \quad (3.5) \]

In the next board all the parameters obtained with uniaxial tensile tests are summed up.

<table>
<thead>
<tr>
<th>E  [MPa]</th>
<th>( \nu )</th>
<th>Se  [MPa]</th>
<th>Sr  [MPa]</th>
<th>n</th>
<th>K</th>
<th>( r )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0°</td>
<td>68079</td>
<td>0.2652</td>
<td>50.89</td>
<td>155.27</td>
<td>0.2553</td>
<td>246.4</td>
</tr>
<tr>
<td>5°</td>
<td>67158</td>
<td>0.269</td>
<td>50.37</td>
<td>155.55</td>
<td>0.2608</td>
<td>251.8</td>
</tr>
<tr>
<td>15°</td>
<td>72056</td>
<td>0.4211</td>
<td>48.99</td>
<td>140.3</td>
<td>0.2491</td>
<td>220.9</td>
</tr>
<tr>
<td>30°</td>
<td>73956</td>
<td>0.4844</td>
<td>48.07</td>
<td>141.1</td>
<td>0.2491</td>
<td>222.7</td>
</tr>
<tr>
<td>60°</td>
<td>77245</td>
<td>0.5077</td>
<td>47.12</td>
<td>143.62</td>
<td>0.2503</td>
<td>227.2</td>
</tr>
<tr>
<td>90°</td>
<td>67918</td>
<td>0.4268</td>
<td>46.86</td>
<td>146.94</td>
<td>0.26</td>
<td>237.4</td>
</tr>
<tr>
<td>mean</td>
<td>70568.67</td>
<td>0.3824</td>
<td>48.71667</td>
<td>147.13</td>
<td>0.2541</td>
<td>234.4</td>
</tr>
</tbody>
</table>

Table 1 - Mechanical properties determined from the uniaxial tensile tests.

This way the mechanical behavior of the material could be approximated by the Ludwik-Hollomon model, with \( K=334.4 \text{ MPa} \) and \( n=0.25 \), as shown in Picture 6.

The anisotropic properties can be quantified along the orientation in the plane of the material, with the calculation of the planar anisotropy coefficient,

\[ \Delta r = \frac{r_0 - 2r_{45} + r_{90}}{2} \quad (3.6) \]
and the mean anisotropy coefficient,
\[ F = \frac{r_0 + 2r_{45} + r_{90}}{4} \] (3.7)

The variation is presented in Figure 7.

![Figure 7 - Variation of the anisotropy coefficient along the orientation of the plane in Aluminum alloy AA4343 (7.5%) – HOGAL 3551 – 7730 (7.5%).]

### 3.1.2 Material model verification with virtual tensile test

Having the main parameters determined, the next step is the creation of numeric material model. The first step involving finite element analysis was the setup of a virtual traction test in order to explore LS-Dyna material model capabilities to represent the behavior for this anisotropic material. In Figure 8 the specimen in finite elements is displayed and the numeric results presented next come from the element SHELL 1254 in the middle of the specimen.

![Figure 8 - finite element mesh of the uniaxial tensile test specimen.]

The model contains 407 shell elements with a section of 1.6 thickness, using the Belytschko-Tsay element formulation with 2 integration points along the thickness. The simulation recreates the same condition on the experimental uniaxial tests. One of the ends of the specimen is constrained (with the single point constraint (SPC) keywords in LS-DYNA) and the other has a prescribed motion with a velocity of 0.0833 mm/s. The material model keyword used was Hill 3R [4], it’s a anisotropic material model provided by ls-dyna that uses ludwik Hollomon’s empiric equation. The inputs for this material were all determined as in the previous section.

Direction dependent material models in LS-DYNA require the usage of the parameter AOPT, for this case its AOPT = 2 for shell elements, as shown in figure 9 [6].

![Figure 9 - Material referential in a shell element with AOPT=2.]

With AOPT=2 the vector “a” is an input with his global coordinates (a1,a2,a3), so that “c” is the material axis normal to the element plane, and “b” the material axis 90 degrees from “a” [6].

Using the vector a components, we could set up models for specimens at 0º (a1=1), 45º(a1,a2=1) e 90º(a2=1), in order to find the effects of rolling direction definition in LS-DYNA.

Also to verify if the Hill 3R model was predicting properly the stress’s in all directions, an analytical approximation of the tension at 45º e 90º was done using Hill 3R criterion. Approximating the strain at 45º and 90º based on a reference that was the experimental results at 0º.

The theoretical (Hill), experimental (Dir), and numerical results are presented on Figure 10.
Observing this results, the numerical results were very close to the experimental results has the theoretical approximations using hill criterion. This proves that the rolling direction was properly defined in LS-DYNA, and that this material model can be used in the forming simulations.

4. Finite element model description

The elements used in the forming steps are called by Fully-Integrated Shell elements or Fast Shell elements [5]. It’s a quadrilateral element with 4 nodes, uses 2x2 integration in the shell plane, these are illustrated in Picture 11.

![Figure 11 - Representation of the Fully-Integrated Shell element and its integration theory](image1)

This element was chosen because of its indication to sheet metal forming analysis involving springback calculation at the end of each analysis [9].

The geometry of the tools was provided by JDEUS and they were meshed with the automesher, with a maximum size of 1.5 mm to the side and a minimum of 0.5 mm. These tools are defined as rigid. The blank had initially the same mesh dimensions, but because of the LS-DYNA adaptive remesh capabilities, the most deformed elements have automatically refined to much smaller sizes, 0.09375 mm are the smaller elements at the end of the second forming step.

In Figure 12 the finite element model of the blank and its constraints are shown.

![Figure 12 - Mesh of the blank and its constraints](image2)

The time step is chosen according to Courant stability condition, LS-dyna explicit dynamic solver requires this condition to be respected. As it is shown in this expression,

\[
\Delta t_{\text{Crit}} \leq \min\left( \frac{\Delta L_e}{c_e} \right) = \min\left( \frac{\Delta L_e}{\sqrt{\rho_e \sqrt{E_e}}} \right) = \frac{0.09375}{\sqrt{2.8 \times 10^{-6}}} = 5.9 \times 10^{-7} \text{s} \quad (4.1)
\]

The contact is defined with the keyword *CONTACT_ONE_WAY_SURFACE_TO_SURFACE, its suitable for forming operations, it defines the contact between two components in a master-slave relation [8].

Where only the slave is checked for penetration, so in all cases of contact the blank is alwais the slave. Also in the contact keyword is possible to define Coulomb’s friction coefficient, and a viscious damping coefficient that was used to avoid high frequency oscilations in the contact of components [8].

In the four steps the analysis ends at 1s, and when the forming is complete, Ls-dyna automatically switches to Seamless implicit springback solver. This method to calculate springback in forming analysis only uses the
The forming simulations start with the movement at same speed of the punch and the binder, the blank is placed on top of the elevator that is momentarily fixed until the blank is pushed down the dye. The dye and the elevator limiter are the only components that are constrained in all directions, the remaining can move in the Z axis. The elevator limiter was not part of the tools supplied by JDEUS, but it was created to replicate the end of the course of the punch, with this limiter, the blank can be truly compressed at the end of each step, what is called the mono block effect. To recreate this mono block effect in the simulations, in all steps the punch is ordered to compress an extra 0.05 mm to ensure a full compression of the stamp.

The binder follows a velocity profile until it reaches the blank and then instantly it switches to the force profile, its done this way because if we only apply force to the binder LS-DYNA calculates the speed that the component can react to such force, resulting in a acceleration gain that ends up with binder hitting the blank a the tools at great speeds. So its speed has to be controlled and the transition to force applying should be smooth.

The deformed shape of the 1st step is imported to the 2nd successively until the 4th, including all process variables like stress, strain and thickness.

In the next picture 15 is shown the deformed shape of the 2nd step with the detail showing the usage of the adaptive remesh.

5. Results
In this section the most relevant results are presented, including a detailed analysis of the problematic zone shown in section 3.
5.1 Formability

The first forming step can be treated as a basic rectangular stamping, so the deformation trajectories are quite simple. LS-DYNA can approximate the forming limit diagram (FLD) by the introduction of the sheet thickness and the hardening exponent. With the FLD we can evaluate multiple defect causes [5]. The zones of the FLD are determined by the margins of safety implied by the manufacturer, that are 20% of allowable thickness reduction, they also include 20% safety margin from the forming limit curve, resulting in the following zones:

![Figure 16 - FLD zones and the respective color key.](image)

The FLD results for the step are presented here, for both top surface,

![Figure 17 - FLD after 1st forming step, in top surface.](image)

and bottom surface,

Figure 18 - FLD after 1st forming step, in bottom surface.

From this results there is no evidence of formability problems, accordingly to what happens in the production.

The formability results in all the steps are displayed in the elements in the following picture.

![Figure 19 - Formability results in the four forming steps.](image)

The problematic zone can be seen in detail here:
LS-DYNA successfully detected a chance of crack after the 2nd forming step, as shown section 3. With this, it was decided to analyses a sample of elements in detail near the problematic zone. This sample of elements is displayed in Picture 21.

Being this the 2nd forming step the deformation acquired in this elements is rather complex comparing to the 1st step so in the FLD diagrams presented next, the limit line is not considered because it’s results wouldn’t be reliable without experimental testing to prove its results [10]. Considering the sample of elements of Figure 21, the resultant main strains in the top surface are presented here.

Relatively to the bottom surface the results are much different,

These results lead us to believe that the reason of the crack is due to high plane stress in some elements.

5.2 Reactions

The reactions come from the contact interfaces defined. The evolution of force usage in each component of the 2nd forming step model is shown next:

This is the step that involve the most force from the press. It’s also the step that changes more the shape of the blank. The imposed forces reacted accordingly to what they supposed to.

The results of force in the INP for all steps is summed in Table 2.
The Time evolution of press requirement in each step is presented next:

![Figure 25 - Evolution of required punch force with press displacement.](image)

Note that these forces should not be evaluated quantitatively because the mono-block effect greatly increases the force required by the punch for each 0.01mm of displacement. This way it was decided to try and quantify the effect of the mono block in each step. By using 2 points of the reaction of the punch in the end of each step we could quantify the slope. With this we have quantified a rate of mono block force for each step.

<table>
<thead>
<tr>
<th>Step</th>
<th>ForceApplied (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1st</td>
<td>80 695</td>
</tr>
<tr>
<td>2nd</td>
<td>193 292</td>
</tr>
<tr>
<td>3rd</td>
<td>104 413</td>
</tr>
<tr>
<td>4th</td>
<td>73 376</td>
</tr>
</tbody>
</table>

Table 2 - Force applied in the closure of the progressive tool in the forming steps. [N]

The variation of force in the closure of the progressive tool isn’t the same for all the steps. Effectively, we can conclude that the rate of mono block force is geometry dependent and the dimensions of contact area significantly change the value of the max force used in each station.

5.3 Tool geometry change suggestion

Confronted with all this results, as an attempt to try to fix the problem happening in production, a simulation of the 2nd forming step was done with a bigger dye corner radii, it was changed from 1.5 mm to 2 mm.

This simulation resulted in a max punch force of 168 KN, which is less, the strains in the problematic sample of Figure 21 in the bottom surface had much lesser elements with high plane deformations, and the results are presented next.

![Figure 26 - Main strains diagram in the bottom surface of the sample of problematic elements, with dye corner radius changed to 2mm.](image)

6. Conclusions

The material model used in numeric models in the four steps shown to be adequate to reproduce the experimental tensile tests that were done.

The simulations allowed to estimate the evolution of punch force required by the sheet metal forming. The end of each simulation reproduced, the closure of the tool along with its big changes in the required force.

The formability results successfully detect the probability of cracks in the expected zone. The change in the dye corner radius, resulted in 13.2% change in total required force, and revealed lesser strains in the problematic elements. The geometric change should be considered if it doesn’t conflict with the existent production line.

As future work, it would be interesting to do same kind of work for all the other stations, so that max press requirement point is known in every step, and to know its influence in moments and forces that could lead to less tool life, to find possible defects, as well as a full study of the formability limits in each step.
7. References