Comparison of hot and cold stamping simulation of Usibor 1500 prototype model

Hande Güler* & Reşat Özcan
Department of Mechanical Engineering, Faculty of Engineering and Architecture, Uludag University, 16059 Gorukle-Bursa, Turkey

Received 1 April 2013; accepted 27 June 2014

Nowadays, high strength steels are being increasingly used in the automotive industry and hot stamping of high strength steel plays an important role of light weighting without affecting the structural performances of final products. Unlike the cold stamping process, blank is formed and quenched, simultaneously during the hot stamping process. This affords the opportunity to manufacture components with complex geometric shapes and high strength. In this paper, the hot and cold forming simulations of a prototype model are processed with Usibor 1500 steel. The forming simulations are performed by DYNAFORM. For this purpose, tensile tests are implemented at elevated and room temperatures. Mat 106 and Mat 18 material models are selected for the hot and cold stampings, respectively. The results from the simulations show that the hot stamping process is safer than cold stamping.

Keywords: Hot stamping, Finite element method, Usibor 1500, Simulation

Recently, sheet metal manufacturing has become one of the most important technologies in automotive industry and stamping is the main process of manufacturing auto-body panels. However, it is very difficult to predict the stamping results because of the quite complex geometry. Moreover, if the mould elements and the process are not planned reasonably, sheet products will be produced with some defects, such as crack, wrinkle, or distortion of blank. Therefore, FE simulation technology is used to predict the defects before real manufacturing. In automotive industry, there are many simulation CAE systems, e.g., Ls-dyna3D, Dynaform, Autoform, and so on. Furthermore, cold (conventional stamping) or hot stamping simulations are performed by the means of these programs.

Hot stamping is a relatively new technology dedicating high strength steel stamping forming and has become the top technology in automotive manufacturing field. High strength steels are used for the hot stamping process and the hot stamping of high strength steels offers the possibility of fuel saving by weight reduction and enhances passenger safety due to its higher strength. Nevertheless, finite element simulation is becoming an increasingly important tool in the process design for structural automotive components to be manufactured using the hot stamping.

Hot stamping is a non-isothermal sheet metal forming technique where the blank is heated at about 900°C, held there long enough and then placed between cooled dies, formed and quenched simultaneously. The current material, which is used by the hot stamping process, is Usibor 1500 boron steel with an aluminium-silicon (Al-Si) layer.

Many studies have been carried out to study numerical simulation of the hot stamping process and defects encountered after forming, such as springback, wrinkling and warping. The cooling effect of hot-stamping dies was simulated by Lei et al. CFX software was used and the define method of contact thermo resistance was presented in the process of simulation. Liu et al. developed a 3D elastoplastic coupled thermo-mechanical finite element model of ultra high strength sheet metal hot forming. Mechanical properties of the material were obtained from unaxial tensile test at elevated temperatures. In addition, hot stamping experiments were conducted and combined with the simulation for the square-box-shaped component. A material model under hot stamping condition of quenchable steel was set up by Xing et al., and the numerical simulation to the whole hot stamping process of hot forming was made with Abaqus software. Hot stamping of aluminium alloy was studied by Zhou et al. The finite element model is set up, and the forming quality is investigated, both numerically and experimentally. The hot stamping simulation was carried out using the isothermal stress-strain behavior of the low alloy steel sheet at elevated temperature. A mini-sized B-pillar part was used as the simulation model. Yanagimoto et al. studied the springback behavior of high-strength steel after hot and warm sheet, both experimentally and numerically. The amount of

*Corresponding author (E-mail: handeguler@uludag.edu.tr)
springback was predicted by elasticplastic FEM. Malek et al. simulated the hot stamping process experimentally and numerically on a laboratory scale. Simulations were performed using the FE program Abaqus. The predicted press force and temperature values were compared to experimental results in order to verify the used FE-model. Tekkaya et al. investigated the finite element simulation of a hot stamping process by means of experimentally calculated material, aiming at a notable decrease in computation time. By this method, the thermal and mechanical phenomena are decoupled and analyzed in two FE-models. Shapiro studied the finite element modeling of 22MnB5 ultra high strength steel. Numisheet 2008 benchmark problem was simulated and LS-Dyna material models were used for the blank, the simulation results were also compared to the experimental data. Fu et al. compared the cold and hot stamping simulations of aluminum B-pillar model. Orthotropic material model Hill90 was used for the simulations, the final shape and thinning of the blank were compared depending on the thickness distribution. Hu et al. studied the hot forming simulation of typical U-shaped steel which was made from 22MnB5. Hot forming simulation was developed as multi-field coupled by commercial software KMAS (King-Mesh Analysis System) and consist of stamping and quenching processes. Temperature distribution and fraction of martensite were obtained by the help of simulations. In other part of the book, hot stamping simulations were studied as two and one-step, blank thickness were compared to experimental results. Dongbin et al. made simulation the hot stamping process of a U-channel with 22MnB5 boron steels. Hot stamping simulation was simulated by using a coupled thermo-mechanical FEM program (LS-DYNA) and temperature distribution was calculated and compared to the measured results. In addition, a program was developed to predict the final microstructure and mechanical properties of the blank. Ertürk et al. implemented a thermo-mechanical-metallurgical model in AutoForm in order to understand the thermal, mechanical and metallurgical effects of the hot stamping process. The implementation was validated by means of hybrid studies of experiments and simulations.

In this paper, the cold and hot stamping simulations of a prototype model were processed with Usibor 1500 boron steel. Dynaform software was employed to build the stamping models of the prototype part. The distributions of Von Mises stress, thickness, and thinning values were obtained by the means of the cold and hot stamping simulations. The result shows that the quality of the part in the hot stamping is better than in the cold stamping.

### Simulation Parameters

In this paper, comparison of the hot and cold stamping simulations of Usibor 1500 boron steel was studied. The simulations were conducted using dynamic explicit FEA software eta/Dynaform. Mat_Power_Law_Plasticity material model was applied to the sheet metal for the cold forming simulation. Mechanical properties of the as-delivered material used in the cold forming simulation are measured by tensile tests at room temperature given in Table 1. Cowper-Symonds coefficients \((C\text{ and } p)\) and Poisson's ratio values for room temperature taken from the literature are given in Table 2.

Hot stamping simulation is a coupled thermo-mechanical analysis, the effect of plasticity increasing and internal energy changing caused by temperature variation must be considered. Stress-strain behavior at elevated temperatures is needed because of the important effect of temperature on the mechanical property.

Mat Elastic_Viscoplastic_Thermal (Mat_106) model was applied to the sheet metal for the hot stamping simulation. This model is implemented in the LS-DYNA code and suitable for coupled thermo mechanical simulations. The most important characteristic of the model is that given \(\sigma\text{-}\varepsilon\) data as a function of temperature at a specified strain rate (Fig. 1). Hot tensile tests are performed to determine the mechanical properties of the steel in dependency of temperature (400, 500, 600, 700 and 800°C) and at a constant true strain rate (0.083 \(s^{-1}\)). Stress-strain

### Table 1—Mechanical properties of the investigated Usibor 1500 steel

<table>
<thead>
<tr>
<th>Material</th>
<th>Yield Strength ((MPa))</th>
<th>Young Modulus ((GPa))</th>
<th>(n)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Usibor 1500</td>
<td>415</td>
<td>222</td>
<td>0.11</td>
</tr>
</tbody>
</table>

### Table 2—Some properties of the investigated Usibor 1500 steel

<table>
<thead>
<tr>
<th>Temperature ((^\circ C))</th>
<th>(C)</th>
<th>(p)</th>
<th>(v)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Room temperature</td>
<td>6.2 (10^9)</td>
<td>4.28</td>
<td>0.284</td>
</tr>
<tr>
<td>100</td>
<td>8.4 (10^5)</td>
<td>4.21</td>
<td>0.286</td>
</tr>
<tr>
<td>200</td>
<td>1.5 (10^4)</td>
<td>4.1</td>
<td>0.289</td>
</tr>
<tr>
<td>300</td>
<td>1.4 (10^3)</td>
<td>3.97</td>
<td>0.293</td>
</tr>
<tr>
<td>400</td>
<td>25.8</td>
<td>3.83</td>
<td>0.298</td>
</tr>
<tr>
<td>500</td>
<td>78.4</td>
<td>3.69</td>
<td>0.303</td>
</tr>
<tr>
<td>600</td>
<td>35.4</td>
<td>3.53</td>
<td>0.31</td>
</tr>
<tr>
<td>700</td>
<td>23.3</td>
<td>3.37</td>
<td>0.317</td>
</tr>
<tr>
<td>800</td>
<td>22.2</td>
<td>3.21</td>
<td>0.325</td>
</tr>
</tbody>
</table>
curves are shown in Fig. 1. These curves are used for determining yield stress and Young's modulus values indicated in Fig. 2. On the other hand, it must be said that the strength decreases as the temperature increases as expected and the formability of the material is improved with the increase of temperature.

The viscous effects of the model for hot stamping can be accounted for by using the Cowper-Symonds coefficients $C$ and $p$, and the flow-stress data can be scaled by $(1 + \frac{\varepsilon}{c})^{1/p}$ for a specified strain rate.

$C$ and $p$ are the Cowper-Symonds coefficients and Poisson's ratio values are taken from literature and are given in Table 2. On the other hand, Thermodynamics parameters of the blank and tools are taken from literature and are given in Table 3. Coefficient of thermal expansion of the blank $\alpha$ was taken from the Ref. as $1.3e-05 (1/\degree C)$ and phase transformations are not investigated in this work.

The blank in hot stamping process is usually formed in a temperature range of 600-800$\degree C$. For that reason, initial blank temperature was determined 810$\degree C$ after transfer to press, and then different temperature values (600$\degree C$ and 700$\degree C$) and were attempted to reveal the effects of hot stamping process.

The length, width, and thickness of blank are 642 mm and 264 mm, 1.7 mm. The geometric model was shown in Fig. 3. Assembly of the geometric model consists of punch, die, blank and binder. Belytschko-Tsay thin shell elements are used for the blank, the tool components are treated as rigid bodies. The fix binder gap is used to hold the blank.

Hot stamping simulation includes closing stage and drawing stage that is the main forming process. The specific conditions for simulation are as follows:

(i) blank holding force was 100 kN, stamping velocity was 6800 mm/s in the cold stamping condition, and
(ii) blank holding force was 100 kN, stamping velocity was 6800 mm/s in the hot stamping condition when the initial temperature of the die, punch and the blank was respectively 75$\degree C$ and 810$\degree C$.

**Results and Discussion**

During a forming process, a sheet metal necks and localizes progressively until it fails when critical limit strains are reached. Forming limit diagram (FLD) is one of the important tools to estimate formability of the sheet metal and the diagram enables that whether deformation can lead to the failure of it. Therefore, the final shapes and forming limit diagram (FLD) of the blanks are shown in Fig. 4. According to Fig. 4a there was an important region in edge section of the cold stamped part that was exposed to crack indicated as red color. On the other hand, the crack didn’t appear in the hot stamping (Fig. 4b-d). The FLD results showed that the part in the hot stamping was safer to manufacture and the hot stamping formability of the blank was better without any crack and excessive thinning phenomenon.

Moreover, Von Mises stress distributions on the formed blank are given in Fig. 5. Maximum Von Mises stress value after cold stamping simulation was about 610 MPa which was shown in FLD diagram as cracked zone (Fig. 5a). This is expected behavior as a consequence of failure. The maximum Von Mises stress value of the hot stamping (810$\degree C$) was about 92 MPa, close to the radius of the blank (Fig. 5b). On

![Fig. 1—Stress-strain curves of Usibor 1500 at elevated temperatures](image1)

![Fig. 2—Yield strength and Young’s modulus values of Usibor 1500 at elevated temperatures](image2)
the other hand, Von Mises stress distributions for the temperatures of 700°C and 600°C are given in Fig. 5c and Fig. 5d and the values were about 125 MPa (700°C) and 136 MPa (600°C). Moreover, these values were lower than the cold stamping. The results showed that the cold stamping process caused higher stress value, which was about four times larger. Therefore, the maximum Von Mises stress within the blank is already well beyond the yield stress, as expected.$^{24}$

The blank thickness variation and the thinning distribution after the cold stamping are shown in Fig. 6. The minimum thickness value after cold stamping simulation reached 1.391 mm (Fig. 6a). Furthermore, the minimum thinning rate was 18.181% indicated in Fig. 6b. According to the figures, wrinkling occurred on cracked zone and caused increasing of the sheet thickness by means of deformation. The maximum thinning rate and minimum thickness after hot stamping simulation (810°C) was 15.196% and 1.442 mm, respectively (Fig. 7a-b). Figure 7b shows that the maximum thinning of the blank after hot stamping simulation occurred in radius zones.

On the other hand, the minimum thickness values for the temperatures of 700°C and 600°C are given in Fig. 8a and Fig. 9a as 1.454 mm and 1.480 mm,

---

Fig. 3—Schematic illustration of tools and blank used within cold and hot-stamping processes (a) punch, (b) die, (c) blank, (d) binder and (e) assembly of the model
Fig. 4—Forming limit diagram of (a) cold stamping, (b) hot stamping process (810°C), (c) hot stamping process (700°C) and (d) hot stamping process (600°C)
Fig. 5—Von Mises Values of (a) cold stamping, (b) hot stamping process (810°C), (c) hot stamping process (700°C) and (d) hot stamping process (600°C)
Fig. 6—(a) Thickness values of cold stamping and (b) thinning values of cold stamping

Fig. 7—(a) Thickness values of hot stamping (810°C) and (b) thinning values of hot stamping (810°C)
Fig. 8—(a) Thickness values of hot stamping (700°C) and (b) thinning values of hot stamping (700°C)

Fig. 9—(a) Thickness values of hot stamping (600°C) and (b) thinning values of hot stamping (600°C)
respectively. The results showed that, as the initial temperature of the blank increases, the minimum thickness value decreases. In Fig. 8b and Fig. 9b, the maximum thinning rates for the temperatures of 700°C and 600°C are also given as 14.492% and 12.940%, respectively. Besides, there exists a decreasing trend in the maximum thinning rate from 810°C to 600°C.

Thickness and thinning results indicated that the deformation of hot stamping was more uniform than that of the cold stamping. According to the all results, hot stamping increases the minimum thickness of the blank and decreases the maximum thinning rate and Von Mises stress. Since, increasing temperature improves the ductility of the material, the blank is formed easier and the minimum thinning increases. On the other hand, Fig. 2 shows that the yield strength and the Young’s modulus decrease with increasing temperature. The main reason for this is that an increase in the mobility of dislocations occurs with increasing temperature. This is because the mobility of dislocations increases as the temperature increases. In brief, the ductility increased as the temperature is increased\(^{25}\). Because of this reason, the formability in the hot stamping process is improved and crack may not occur.

In addition, the cold forming causes a decrease in ductility and if the cold forming exceeds certain limits, the crack occurs on the blank material before reaching the desired shape. Therefore, for the purpose of avoiding such difficulties, the cold stamping processes are usually carried out in several steps or the blank material may be softened and so, the ductility is restored\(^{26}\). For this reason, the cold stamping simulation showed that the formability in the cold stamping process is limited and the ductility is of major importance in forming process.

Conclusions

For Usibor 1500 steel, flow curves at elevated and room temperatures were obtained from experiment and other needed properties were taken from literatures. The hot and cold stamping simulations were carried out by Dynaform software. The cold forming process caused the failure of the blank. Because, the blank was more brittle and it was more difficult to form it during the cold stamping process. To overcome forming problem, hot stamping was introduced to the blank. The result shows that the hot stamping of the Usibor 1500 material could meet performance requirements of a forming process without cracking. Compared with the cold stamping, more appropriate values of the thickness-thinning distribution, the forming limit and the Von Mises stress was obtained for the hot stamping process. In addition, it is proved that the hot forming simulation method was correct for real manufacturing process and could be used to predict forming limits before real manufacturing process.

References

18. Erturk S et al., Simulation of Tailored Tempering with a Thermo-Mechanical-Metallurgical Model in AutoFormplus paper presented at 8th Int Conf Workshop on Numerical