Deep drawing of press hardening steels

M Tomasch, W Ecker, R Kaiser and T Antretter

1 Institute of Mechanics, Montanuniversitaet Leoben, Franz Josef Straße 18, Leoben, AUT
2 Material Center Leoben Forschungs GmbH, Roseggerstraße 12, Leoben, AUT

E-mail: melanie.tomasch@unileoben.ac.at

Abstract. Press hardening of low alloy steel sheets is an efficient way of manufacturing high strength automotive components relevant for passenger safety in case of a crash. The press hardening process combines the advantages of good formability at elevated temperatures as well as high strength and good form accuracy after quenching in the closed die. Since cooling during forming and subsequent quenching has a significant influence on the mechanical behaviour of a steel sheet, a strongly coupled thermo-mechanical model has been developed for the simulation software ABAQUS. Accurate simulation results require the knowledge of temperature dependent thermal and mechanical material properties as well as occurring interactions between the sheet and the die. They were determined in a series of experiments and provide the input for the numerical model. During the quenching step in the closed die, the high strength of the component is achieved by a martensite phase transformation. This transformation is strongly influenced by the deformation in the heavily bent areas as well as the temperature evolution of the sheet. Accurate results for the shape of the final components and for the properties within the operating service time can only be obtained if the numerical model accounts for the microstructure formation including the TRIP (Transformation Induced Plasticity) effect. This is accomplished by means of a user subroutine. Consequently, the model allows to analyse the forming and cooling process in the closed die in detail throughout the entire process cycle. Concluding, the final geometry, the wall thickness distribution of the component and the press forces during forming are used as parameters to compare the simulation results with forming experiments and to validate the chosen material model.

1. Introduction

Press hardening enables cost and time efficient manufacturing of form accurate, light weight and high strength components for the body in white of a car relevant in case of a crash. During press hardening low alloy steel sheets are hot formed and quenched in the same die. Due to the elevated temperature during forming, the required press force is reduced and the formability of the material is enhanced. Furthermore the martensite phase transformation occuring during quenching reduces spring back and increases strength which in turn allows to reduce the sheet thickness of the component.

At the beginning of the direct press hardening process, the sheet metal is heated in the oven from room temperature with ferrite-pearlite microstructure up to approximately 900 °C, where full austenitization occurs. During the subsequent transfer from the oven to the press there is a heat flux due to convection and radiation to the environment. The still austenitic sheet is placed into the water-cooled die and the final form of the component has to be reached at temperatures higher than the martensite start temperature in one single stroke of the press. When the specimen has reached its final shape, it is cooled down rapidly in the die with rates higher than approximately 27 °C/s for a 22MnB5 steel according to [1], to enforce a fully martensitic microstructure. The cooling rates are crucial to the final microstructure and thus the final mechanical properties of the part in service. The phase transformation from austenite to full martensite
leads to a significant increase in tensile strength up to 1500 MPa according to [2,3]. By means of direct hot forming complex, crash resistant parts such as bumpers, pillars and side impact beams with ultrahigh strength, minimum spring back, and reduced sheet thickness can be produced.

2. Cooling Experiments
For the simulations described subsequently it is important to know the temperature at the beginning of the forming process defining the initial stress free state. To this end cooling experiments with rectangular press hardenable steel sheets were conducted. A thermocouple was attached centered on the upper side of the sheet which allowed measuring the temperature evolution during oven dwell time and transfer to the press. During cooling the temperature was additionally measured by pyrometers. Figure 1 shows the difference between the thermocouple and pyrometer measurements. It can be seen that the temperature measured by the thermocouple is slightly higher than the pyrometer measurement. Eventough the difference is not large it has to be considered, since it is absolutely crucial to predict the start of the phase transformation in the subsequent forming simulation accurately. It is not possible to measure the temperature by thermocouples during the forming experiments described in section 4. Therefore the temperature of the sheet during the transfer to the press is measured by the same pyrometers used during the cooling experiments and has to be adapted by adding the known deviation of the pyrometer and thermocouple measurements. The time of transfer varies according to the desired forming start temperature. By means of these experiments the exact temperature at the beginning of forming could be obtained as input parameter for the simulation.

3. Cooling Simulation
The cooling process was simulated in a finite element heat transfer analysis. Due to the symmetry only one quarter of the sheet had to be modeled. The temperature dependent heat transfer coefficients of free convection were calculated by means of dimensionless numbers in particular the Nusselt and Rayleigh number. The emissivity value was set constant throughout the cooling simulation according to the emissivity setting of the pyrometer during the cooling experiments. The previous measured cooling curves were used to validate the chosen input parameters of free convection and radiation used in the numerical model. The measured and simulated temperature evolutions during cooling were compared to each other. The results showed already good agreement to each other as seen in figure 2.

![Figure 1. Comparison of the thermocouple and pyrometer temperature measurements during cooling under atmospheric conditions.](image1)

![Figure 2. Comparison of measured and simulated temperature evolution during cooling under atmospheric conditions.](image2)
4. Forming Experiments
Forming experiments of a simple hat shaped profile were conducted to allow a validation of the numerical forming and press hardening model by comparing the final shape and wall thickness distribution. During the experiment a presshardenable steel sheet was austenitized in the oven. After complete austenitization a robot removed the steel sheet from the oven and transferred it to the die. During this process the sheet further cooled down due to radiation and convection. Shortly before the sheet was placed into the die, the temperature was measured by pyrometers. High tool velocities were required to ensure forming to final shape in one stroke in a still austenitic state. After forming the cooled die remained closed until completion of the martensitic phase transformation in the specimen. After removing the specimen from the die it cooled down to room temperature at atmospheric conditions and the wall-thickness was measured at different positions along the cross section. Three different forming start temperatures were chosen to study their influence on the phase transformation and the final specimen shape.

5. Forming Simulation
The simulations were performed using the commercial software package Abaqus/Standard. According to [2,3] a strongly coupled temperature-displacement analysis is required due to the fact that contact between the die and the sheet during forming and press hardening significantly influences the temperature evolution.

Due to the shape of the formed profile a 2D plane strain model representing the cross section of the hat profile sufficed to study the principle thermo-mechanical response. To reduce the calculation time only one half of the symmetric hat profile was simulated. Since the heat flux from the sheet to the die during forming and especially quenching must not be neglected all tools were modeled as elastically deformable parts with temperature degree of freedom and an initial temperature of 25°C. The model contains three tool parts, the die, the punch and the blank holder, see Figure 4. The sheet of the numerical analysis had the same shape as the blank used for the forming experiments. The tool’s thickness was chosen to avoid plastic deformation and to restrict the temperature increase only to the sheet contact region.

For the material description the flow curves were available at various temperatures within the relevant interval from room temperature to temperatures just above austenitization. For implementation into Abaqus/Standard classical Mises plasticity with isotropic hardening was chosen. Also the interaction properties, i.e. the friction coefficient as well as the heat transfer coefficient across the contacting bodies are temperature dependent as investigated by [4,5]. Note that the latter is also a function of the contacting pressure. This effect had to be taken into account in the model.

The simulation was divided into 3 steps. During the first step the sheet with an initial temperature between 500°C and 700°C was formed in one single stroke. The simulations accounted for all sources of non-linearities, i.e. large displacements, material non-linearities, as well as contact and friction. The process parameters such as drawing depth and tool velocity were set in accordance to the experimental process parameters. In the second step the tools remained closed for 5 seconds and the sheet cooled down due to the heat flux to the die. In the third step the tool contact was deactivated and the specimen cooled down to room temperature under atmospheric conditions.

The two dimensional solid elements CPE4RT with four nodes, displacement and temperature degree of freedom and reduced integration were used for the tools as well as the specimen. A fine mesh with an element length of 0.5 mm and 10 elements evenly distributed along the sheet thickness was used for the specimen to enable smooth bending. A coarser mesh was used for the tools with the exception of the tool radii. A finer mesh was applied to the curved tool contours to ensure a smooth radius and to avoid unrealistic forming conditions due to the element discretization.

5.1. Phase Transformation
In order to account for the phase transformation during quenching and its influence on the final shape and mechanical properties of the formed component, an accurate thermo-mechanical metallurgical model had to be provided. The model considers the interaction between the mechanical field induced for
example by forming and thermal expansion, the thermal field evoked by phase transformation, plastic forming and friction, as well as the microstructure evolution in dependence on the temperature and the stress-strain state. Moreover the influence of the microstructure on the mechanical and thermal properties of the material is considered. The model’s functionality was adapted to this particular analysis requirements by taking advantage of various subroutines available in Abaqus/Standard.

The user subroutine USDFLD was used to calculate the phase fraction of martensite $\zeta_M(T)$ following a slightly modified kinetic law of Koistinen and Marburger used by [2,6]. If the simulated temperature is below the martensite start temperature $M_s$ and the cooling rates are high enough, martensitic transformation occurs according to

$$\zeta_M(T) = 1 - e^{-\alpha(M_s-T)^n}$$

where $T$ denotes the current temperature and $\alpha$ as well as $n$ are material parameters. In this specific case the martensitic phase fraction is stored as a field variable. The material properties are then made dependent on this field variable. The field variable along with some other state variables, updated in the USDFLD routine, are passed into the user subroutines UEXPAN, CREEP and HETVAL that are also called at the integration points. Subroutine UEXPAN is used to define the incremental thermal strains as function of temperature and state variables representing the phase contents. In this way the shrinking during cooling as well as the volume change due to the phase transformation are considered. State variables updated by UEXPAN are passed into user subroutine CREEP which accounts for transformation induced plasticity (TRIP) effect. The austenite to martensite phase transformation influences also the temperature evolution due to the latent heat release during phase change. To this end the values of state variables, already updated by USDFLD, UEXPAN and CREEP subroutine, are finally passed into the HETVAL subroutine, to define the heat flux due to internal heat generation in the material as it occurs during the phase change. A comprehensive description of the interplay of the subroutines necessary for capturing the effects of martensitic phase transformation and implementation details can be found in [7].

6. Results
The simulation results were compared to experimental results to validate the chosen input parameter. In a first step the wall thickness of the specimens formed during the experiments were measured at five different positions distributed along the specimen’s half cross section. In a second step the flange width of the hat shaped profile was measured. The positions of the wall thickness measurement as well as the location of the flange width measurement are shown in figure 3. Due to the symmetry only one half was simulated and compared to the experimental results. As it can be seen in table 1 the wall thickness as well as the flange width predictions already show a good accordance with the measurements.

| Position | 1 | 2 | 3 | 4 | 5 | Flange width | Temperature |
|----------|---|---|---|---|---|--------------|-------------|
| Simulation | 1.52 | 1.38 | 1.37 | 1.38 | 1.52 | 28.0 mm | 500°C |
| Experiment | 1.53 | 1.39 | 1.38 | 1.38 | 1.50 | 28.9 mm | 500°C |
| Simulation | 1.51 | 1.38 | 1.38 | 1.38 | 1.50 | 27.9 mm | 580°C |
| Experiment | 1.52 | 1.39 | 1.40 | 1.39 | 1.49 | 28.6 mm | 580°C |
| Simulation | 1.51 | 1.38 | 1.38 | 1.37 | 1.49 | 27.6 mm | 670°C |
| Experiment | 1.51 | 1.40 | 1.39 | 1.41 | 1.48 | 27.8 mm | 670°C |
This is an indication that the mechanical forming behaviour as well as the forming temperature and friction between the tool and the sheet have been calculated correctly. Since the simulation results for the wall thickness and flange width already showed satisfying agreement with the experimental evidence, the press force which was recorded during the forming experiment has been compared to the simulated tool’s reaction force to further investigate the reliability of the simulation. A validation of the required press forces is important since it also indicates that the plastic forming behavior, specimen’s temperature and friction has been chosen correctly as input parameter for the numerical model. Since it is challenging to predict the exact conditions during forming especially regarding the friction coefficient and the thermal conductance in dependence of surface clearance and contact pressure, 4 variants were simulated using two different friction coefficients and two sets of thermal conductance values whereby letter L indicates the lower and letter H the higher values of the varied parameters. The simulated as well as measured press force during forming is depicted in figure 5.

According to figure 5 the higher friction coefficient and higher thermal conductance between tools and sheet led to a press force closest to the recorded press force during forming.
7. Conclusions
A strongly thermo-mechanically coupled simulation of the press hardening process of a simple hat shaped profile was developed. Cooling experiments and simulations were conducted to determine the heat transfer parameters of convection and radiation during the transfer to the die. A series of experiments was required to investigate the material and contact properties used as input data for the subsequent forming simulation. Abaqus/Standard user subroutines were implemented in the simulation to account for the martensite phase transformation during quenching. The simulation results were compared to the experimental results of the press hardening process and the used material, interaction and process parameters were successfully validated.

Acknowledgments
Financial support by the Austrian Federal Government (in particular from Bundesministerium für Verkehr, Innovation und Technologie und Bundesministerium für Wissenschaft, Forschung und Wirtschaft) represented by Österreichische Forschungsförderungsgesellschaft mbH and Standortagentur Tirol, within the framework of COMET Funding Programme is gratefully acknowledged.

References
[1] Karbasian H and Tekkaya A E 2010 Journal of Materials Processing Technology 210 pp 2103–2118
[2] Behrens B A et al 2007 Numerical simulation of microstructure evolution during the hot stamping process (Gyor, Hungary: IDDRG Conf.)
[3] Abdollahpoor A et al 2013 Applying a modified austenite transformation model into a thermo mechanical model of hot stamping (Zurich Switzerland: IDDRG Conf.)
[4] Lechler J and Merklein M 2008 Hot stamping of ultra high strength steels as a key technology for lightweight constructions (Pittsburgh, Pennsylvania: Material Science and Technology)
[5] Hochholdinger B 2011 Simulation of the press hardening process and prediction of the final mechanical material properties (Seoul, Republic of Korea: Numisheet Conf.)
[6] Blaise A et al 2013 Applied Thermal Engineering 61 pp 141–148
[7] Antretter T et al 2010 Steel Research International 81 pp 675–680