Press-Hardening Simulation - the next Level of Maturity

T Schönbach\textsuperscript{1}, M Messner\textsuperscript{2} and C Gmainer\textsuperscript{3}

\textsuperscript{1} Technical Account Manager, AutoForm Engineering Deutschland GmbH, Pfaffenhofen a.d.Ilm, DE
\textsuperscript{2} Development Engineer, AutoForm Development GmbH, Zürich, CH
\textsuperscript{3} Technical Sales, weba Werkzeugbau Betriebs GmbH, Dietach, AT

e-mail: thomas.schoenbach@autoform.de

Abstract. Press hardening has been fully established in the automotive industry during the last years. Forming simulation is an important tool to make the whole potential of press hardening available in both, the mass production and further developments. One of the usual simulation packages is AutoForm. It has been known as a reliable software tool for the sheet metal forming application area as well as for the press hardening for years. AutoForm-ThermoSolver\textsuperscript{plus} supports direct and indirect hot forming, which are followed by quenching and cooling processes. It takes into account phase transformation during quenching and thermal distortion after cooling. The recent version AutoForm\textsuperscript{plus} R7 brings further advances in process modeling for hot forming as this release allows users to take into account cooling channels. Based on this, the heat conduction in tools can be calculated. Cooling channel modelling affects press hardening simulation in two ways: it can be used to optimize the cooling channel design in the press hardening tools and it increases the accuracy level of the simulation in general. The resulting benefits coming from these new features will be presented in this paper; based on a B-pillar which was produced with direct press-hardening process.

1. Introduction
Hot forming with cooled tools is state of the art for the production of structural parts made of ultra-high strength steel in the automotive industry. As the final part properties depend on the cooling process, it is extremely important to appropriately design the tools and their cooling systems. Therefore the current AutoForm\textsuperscript{plus} R7 release allows the user to model them. Additional to that numerous improvements of material and process modeling have been integrated in the AutoForm-ThermoSolver\textsuperscript{plus} during recent developments. Herewith direct and indirect press-hardening processes can be more accurately represented in the simulation. The following describes the essential innovations more detailed.

2. Material Model
The detailed material description over all phases of the press-hardening process is an essential innovation of the AutoForm-ThermoSolver\textsuperscript{plus}. Starting with the heating of the blank in the furnace, over the transport of the blank to the tool, the forming in the tool and the final quenching in the tool as well as the cooling down to ambient temperature outside the tool.
2.1. Temperature Dependent Young’s Modulus

The Young’s Modulus of the material 22MnB5 is temperature dependent. It amounts to 210,000 MPa at ambient temperature, while it only amounts to 45,000 MPa at 950°C. Between these two temperatures Young’s Modulus is linearly interpolated. Using the temperature dependent modelling the blank behavior can be very precisely simulated, e.g. the insertion behavior of the hot blank. For higher temperatures the smaller Young’s Modulus ensures a greater amount of blank sag considering the influence of gravity during the blank insertion into the tool.

2.2. Phase Dependent Material Behavior

The AutoForm-ThermoSolverplus describes each phase both temperature and strain rate dependent. That means that each of the phases, such as bainite, martensite, ferrite-pearlite and austenite is described by a number of temperature and strain rate dependent hardening curves. Thus, the material can be described completely thermo-mechanically coupled at every time of the simulation. Both the thermo-mechanical behavior of each phase and the influence of corresponding phase fraction are taken into account in the simulation. The heating of the blank in the furnace, over the transport of the blank to the press, the blank insertion, the forming in the tool, the quenching in the tool as well as the cooling down to ambient temperature are now simulated completely thermo-mechanically coupled. The phase transformations and the corresponding phase fractions are modeled and described by the GMS model [1].

Figure 1. CCT (schematic) according to [2], A: Austenite, B: Bainite, F: Ferrite, M: Martensite, P: Pearlite

This model is based on measurements which describe the physical material behavior. The chemical composition of the material 22MnB5 is used as input for this model. Thus, this model also allows the simulative description of smaller variations of the chemical composition. The so-called CCT diagram (‘figure 1’) is the basis of the simulation of the phase transformation. According to the current temperature and cooling rate it is checked whether a phase transformation is realized. If so, also the amount of the phase formation rate is determined. The influence of the forming history on the phase transformation is also taken into account by the GMS model. In ‘figure 1’ the translation of the bainite nose to the left for greater plastic pre-deformation is schematically shown. Due to that the critical cooling rate for martensite formation increases, too. Using this phase transformation model the phase fractions can be very accurately determined according to forming history, contact state and, thus, the individual local cooling rate. Based on that the final mechanical part properties are calculated and it can be checked, whether the expected behaviors, are fulfilled for the part.

3. 3D Tool Modeling

Usually in a standard forming simulation tools are described as 2D geometries. But in the current case, for a press hardening simulation including the cooling channels, tools must be described as 3D geometries. For this purpose AutoFormplus automatically generates the 3D tool meshes based on the
existing active forming tool surfaces. During this meshing procedure, firstly the active surfaces are refined in the relevant areas, secondly, the lateral and bottom surfaces are generated and, finally, a 3D mesh is generated (‘figure 2’). Therefore an optimized 3D mesher has been developed and integrated into AutoFormplus R7 in cooperation with CES Eckard GmbH.

In order to ensure that the automatic 3D meshing of both the tools and the included cooling channels is as robust and stable as possible, the cooling channels are not considered by their explicit surface in this 3D mesh. Instead of that, the 3D mesh is locally refined along the cooling channel axis; so the cooling channel surface can be taken into account in the FE analysis with the help of this locally refined mesh (‘figure 2’ left side). In the next step the software automatically identifies the elements which are located inside the cooling channel and assigns them to the locally refined 3D mesh; all the other elements which are located outside the cooling channel are assigned to the 3D mesh of the tools. The idea of this model is to detect volume elements, which lie inside the cooling channel, decouple them from the rest of the mesh, extract and correct their outer surface and apply the convective boundary condition on it. The correction makes sure, that the used ‘cooling effective surface’ approximately corresponds to the geometrical channel surface (‘figure 2’).

![Figure 2. 3D mesh of the B-pillar punch with locally refined cooling channel mesh](image)

To consider the behavior of the cooling channels in the most efficient way, no flow simulation is realized, but the cooling capacity is modeled by a convective heat transfer at the walls of the cooling channels. For that purpose the corrected cooling channel surface as shown in ‘figure 3’ is used. The heat transfer from the tool to the cooling liquid in the cooling channels is represented by the heat flux density

\[
\dot{q} = HTC_{Kk} * (T_{WK} - T_{KF})
\]

(1)

with the convective heat transfer coefficient \( HTC_{Kk} \), the temperature of the cooling liquid \( T_{WK} \), and the local tool temperature \( T_{KF} \). As the entry and exit temperature of the cooling liquid only slightly differs in the real processes, the cooling liquid temperature is assumed to be constant.
4. Coupling with the Forming Simulation

The 3D heat conduction is thermally completely coupled with the forming simulation, in each increment. This enables an accurate simulative description of the real temperature state of the tool:

- Firstly the heat loss from the part to the 3D tools is determined. For that purpose the current thermal contact state, the current part temperature, the current material phases and the current tool temperature are used.

- Secondly the inversed thermal contact information is used to map this heat loss of the part as heat input on the corresponding 3D tools. The heat loss of the part is mapped on the corresponding tool by the contact information and, thus, it is made sure that no energy is lost. Thus, the new tool temperature can be determined.

- Finally the new mechanical state is determined.

As the significant thermal effects result from the coupled simulation, the time integration for the 3D heat conduction in the tool is realized by the implicit Euler method. The additional computation time for the 3D heat conduction in the tools can be minimized by the optimized modeling and 3D meshing of cooling channels, as described in ‘chapter 3’ as well as by the previously described coupling of the thermal and forming simulations – it is less than 20% of the required computation time for the pure forming simulation. ‘Table 1’ shows the computation times for the drawing operation D-20 of the B-pillar with and without 3D heat conduction in the tools (for this example the increase of computation time only amounts to 5%). The number of tetrahedrons in the entire 3D tool set amounts to \( \sim 2.6 \times 10^6 \).

Table 1. Comparison of the computation time for the drawing operation D-20 of the B-pillar with/without 3D heat conduction in the tools

| Simulation                              | Computation Time [sec] |
|-----------------------------------------|------------------------|
| Without 3D heat conduction in the tools | 872                    |
| With 3D heat conduction in the tools    | 917                    |

Figure 3. Locally refined mesh with extracted (left) and corrected (right) effective cooling channel surface
5. Multicycles

Another important aspect during real serial hot forming processes is that the working temperature of the tools is only reached after a number of cycles. The number of these cycles and the real working temperature of the tools are not known in advance. But they must be considered in an accurate hot-forming simulation. In AutoForm\textsuperscript{plus} they can be determined in a so-called Multicycle simulation, using several forming cycles.

5.1. Real Cycling

The principle of Multicycle simulation realized in AutoForm\textsuperscript{plus} will be presented on the example of direct hot forming process (H-10, D-20, K-30). The Heating (H-10) is only considered once, it does not have to be repeated, because there are no tools involved and thus each blank has the same property at the end of H-10 in the serial process. But it is not the same for Drawing (D-20). At the beginning of Drawing (D-20) the drawing tools are cold. They warm up during the forming process. This means that the tools are warmer at the end of the first Drawing cycle. Consequently the initial tool temperature for the second Drawing cycle does not match the first one. Therefore, D-20 must be calculated again, but now with the higher initial tool temperature, resulting from the first D-20 cycle. This procedure must be repeated several times, until the tool temperature at the end of D-20 cycle matches the tool temperature at the end of the previous D-20 cycle. This temperature will be the working temperature of the tools. There are two options for this procedure: Either the number of cycles can be defined and the final temperature of drawing tools is the result; or simulation cycles are rerun until a constant temperature state is established in all tools. Once such stable state is achieved neither part properties nor the tool properties vary in further cycles. Then D-20 operation is finished and the Cooling (K-30) can be simulated. K-30 happens without tool contact. So it must be calculated only once.

This type of Multicycle simulation is called Real Cycling in AutoForm\textsuperscript{plus}, because it provides a realistic view and assessment of the whole real process including also the entry process, the tools as well as part characteristics. Depending on the process, 5-10 cycles are necessary until a constant temperature state in the tools is established.

5.2. Numerical Cycling

There is also another Multicycle simulation method in AutoForm\textsuperscript{plus}, the so-called Quick Numerical Cycling. For that it is assumed that the working temperature distribution in the tools at the beginning of the drawing operation (D-20) is known. This means that Multicycle simulation does not start with cold tool, but with tool of the estimated working temperature.

This estimated working temperature is calculated from all previously known process parameters in the following way: From the energy retained in the initially heated blank a time-averaged heat flux density is determined and projected on the tools in the corresponding shares. This corresponds quite closely to the tool working temperature at the time of blank insertion [3]. Starting with estimated working tool temperature Multicycle simulation is carried out until constant temperature state is reached in D-20 cycles.

Due to this assumption, Quick Numerical Cycling requires only 3-5 cycles to achieve the constant temperature state, instead of the 5-10 cycles in Real Cycling. In this modus the number of cycles is automatically defined to achieve the constant tool temperature state. It should be explicitly mentioned that Quick Numerical Cycling more quickly achieves the constant temperature state for the serial process. The cycles until this state is reached do not represent the real process. Solely the last cycle corresponds to the state which will be established after the entry phase in the serial process.
6. Comparison to practice with the example of a B-pillar
In cooperation with the company Weba various tests were conducted in a test die with different process parameters and blanks. These different processes were simulated with AutoForm-ThermoSolverplus and compared to the test results. As an example some of these comparisons are outlined in the following.

6.1. Temperature histories
As previously described the cooling rate has a major effect on the phase transformation and thus on the part properties. Therefore temperature measuring heads were installed at several positions in the test die. The temperature of the sheet part was measured over time. The goal of the simulative assessment was to determine the parameters of the heat transfer to the air, of the heat transfer between sheet and tool as well as of the gap dependent heat transfer to be able to simulate the temperature histories realistically. ‘Figure 4’ shows the temperature history over time for one measuring point. The white curve describes the ram stroke over time. The press closes during the first 1.3 seconds in rapid motion, forms the part in the following 2 seconds and afterwards keeps the tool closed for a few seconds to harden the part. Due to the design of the used temperature sensor the temperature in the test could only be measured to minimum 400°C. The two black curves show the temperature history of the test and the simulation over the measured time period. They match very well.

![Figure 4. Temperature history in the test and the simulation](image_url)

6.2. Sheet thickness
Sheet thickness distribution is an important simulation result to be able to detect and avoid local necking and possible risk of splits in the part. Thus time and cost intensive change loops can be avoided during the tool spotting. In the described project sheet thickness resulting from simulation was compared to real part. In doing this, parts were digitalized on both sides and using the software ATOS the sheet thickness was projected as color shaded variable on the digitalized data. Additional to this, polished micrograph sections were generated at some positions on the sheet and the sheet thickness was measured again. Herewith ATOS measurements could be double checked. ‘Figure 5’ shows the measured and calculated sheet thickness at the thinnest area of an elliptic embossment. The initial sheet thickness was 1.5 mm. Thus a reduction of sheet thickness of over 30% occurs; which was calculated in the simulation as well.
6.3. Hardness

In order to check the simulation quality regarding the final part properties, such as hardness, tests with different transport and tool closure times were conducted. At several positions of the part the hardness was measured and compared with the calculated values of the simulation. ‘Figure 6’ exemplarily shows the measured and calculated hardness values for 4 measurement ranges. The hardness was measured in two positions per measurement range. Compared with the measured hardness values the simulated hardness values lie within the measurement tolerance of ±20 HV. These results prove that AutoForm-ThermoSolver plus accurately calculates the part properties, if all process and simulation parameters are precisely adjusted.
6.4. Distortion
During this test, firstly the real process was examined, to find out how process parameters influence the distortion. Secondly the simulation was checked, to find out whether the same behavior can be detected in calculations. Since the distortion was very low in real part, a special test was used – the forming process was stopped 10 mm before the bottom stroke. In ‘figure 7’ the upper black curve shows the section through the partially still hot real part in the tool. The lower black curve represents the section through the cooled real part. The curves show that during cooling process part is distorted. The maximum value is on the right side of the part and it amounts to up to 10 mm. The thicker, colored curve represents the section through the simulated, cooled part. In the main part the calculated distortion fits the measured one. There is only one low, locally limited difference between measured and calculated distortion of 1 mm.

![Figure 7. Distortion in the test and the simulation – Distance to experiment in millimeter](image)

7. Summary
New integrated features of the AutoForm-ThermoSolver\textsuperscript{\textcopyright} R7 enable the process engineer to design and optimize all types of press hardening processes. As shown in the example of the B-pillar all required part properties can be validated beforehand. Currently cooling channels have to be designed in CAD. For the future the integration of functionalities for the cooling channel design and optimization are planned into the AutoForm\textsuperscript{\textcopyright} software.

8. Acknowledgement
AutoForm Engineering Deutschland GmbH likes to thank the company Weba, especially Mr. Gmainer and Mr. Graf for the conduct of the practical tests and the provided measured values.

References
[1] http://www.gmt-berlin.com
[2] Merklein M and Svec T 2010 Transformation kinetics of the hot stamping steel 22MnB5 in dependency of the applied deformation on the austenitic microstructure Proc. 50th Conf. of IDDRG 2010 ed R Kolleck (Graz: Verlag der technischen Universität Graz) pp 71-80
[3] Weiss W 2013 Thermische Auslegung von Werkzeugen für Presshärtprozesse (TU Graz)