Coupled Simulation of the Fluid Flow and Conjugate Heat Transfer in Press Hardening Processes

Bruno Boll[1], Uli Göhner[1], Inaki Caldichouri[2]

[1]DYNAmore GmbH
[2]LSTC

Abstract

Due to the increasing demands on lightweight design, stiffness and crash performance of automotive body components, the press hardening method becomes widely-used. The high strength of press hardened parts of up to 1.5 GPa results from the nearly complete conversion of austenite into martensite. This microstructural transformation, also known as 'hardening', happens during or subsequently to the forming process. In order to achieve a cooling rate which is high enough to get a martensitic microstructure in all regions of the blank, it has to be ensured that the heat transfer rate from the blank to the tool and inside the tool is sufficiently high. This is usually achieved through the cooling of the tools with a fluid.

This presentation describes the coupled simulation of the flow through the cooling passages and the temperature distribution in blank, tool and fluid in a complete forming cycle. A completely shaped blank is used just from the beginning of the simulation. The distribution of the heat transfer coefficient along the contact surface between blank and tool is determined beforehand through a thermal-mechanical forming simulation, which is not part of this presentation.

Subject of the current investigation is the simulation of the transient, turbulent and viscous flow with conjugate heat transfer including the determination of the temperature distribution inside blank, tool and fluid and the heat transfer from blank to tool and from tool to fluid using LS-DYNA's® monolitically coupled ICFD and thermal solver. The fluid is assumed to be incompressible with the flow properties of water. The initial temperature distribution in the tool is determined beforehand using thermal-only simulations of multiple consecutive forming cycles where the temperature at the end of one cycle is used to initialize the tool temperature of the subsequent cycle. Simulations with all turbulence models available in LS-DYNA's ICFD solver are performed and the results are compared.

1 Introduction

Press hardening is a standard process for the production of ultra high strength structural steel parts. A thermomechanically coupled solution strategy has to be followed. Prior to the forming process, the blank is heated up to become fully austenitized. Forming takes place completely in the austenitized state. When the temperature is decreasing, the austenite decomposes into one or more product phases. A higher cooling rate increases the amount of the harder phases bainite and martensite.

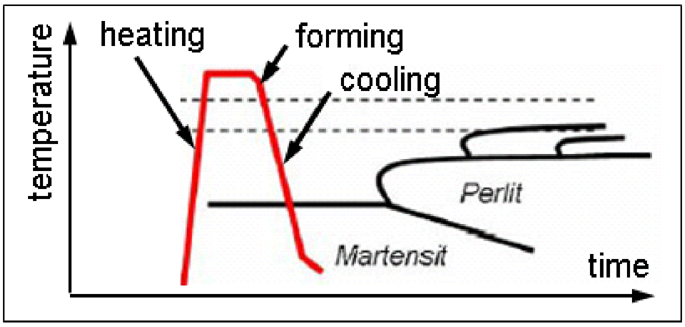

Fig.1: typical temperature profile of press hardening process in TTT diagram

Therefore, to get a particular amount of heat in a particular amount of time out of the blank is an essential requirement in the press hardening process. The cooling of the blank occurs almost completely through the heat transfer from the blank to the tools. Therefore, the cooling of the tools is an essential need and is usually achieved by a water cooling system.

In order to reduce the manufacturing cost per part in the serial production of automotive body components, it is sought to reduce the cycle time as much as possible in order to produce as many parts as possible in a particular time. To ensure to get the desired microstructural material composition, the tools have to keep closed until the temperature in all regions of the sheet is not higher than 150 °C – 200 °C to guarantee that the temperature is well below the martensite finish temperature which lies at about 280 °C [1]. Due to the relationship between the reduction of cycle time and the manufacturing costs per part, there is a vital interest of the manufacturer of press hardened automotive body components in the efficiency of the cooling system of the tools.

2 Simulation Model

2.1 CFD Simulation

The complete tool is divided into four independent segments for the punch and four segments for the die. The transient, turbulent and viscous flow through the cooling ducts of each of the four segments of the punch was calculated using LS-DYNA's ICFD solver. The inlet velocity was varied until a pressure drop of Δp between inlet and outlet is reached. This iterative process is an inverse approach because in the real world problem, the pressure drop is a known and the flow velocity is one of the unknown quantities. This approach is due to the fact that in CFD simulations usually no convergence can be reached with a pressure-pressure boundary condition. A mass flow rate and a pressure have to be provided in CFD simulations instead.

The mesh size and the time step size was determined beforehand using sections of the complete channel. First, a linear pipe was cut out of the complete pipe of punch segment 1. For this kind of problem, an analytical solution (Poiseuille flow) is available. Mesh size and time step size were varied until a good agreement between analytical and numerical solution was reached.

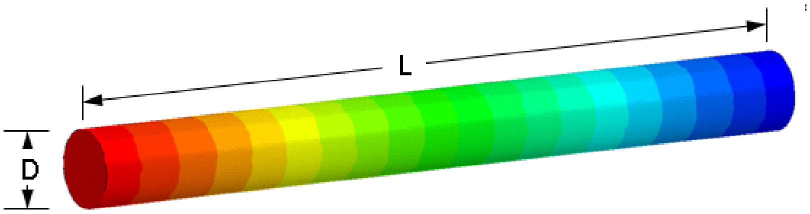

Fig.2: linear section of punch segment 1 with surface pressure (Poisseuille flow)

A second u-bend shaped section was cut out of the pipe of punch segment 1. With this model, the relationship between mesh size, time step size and simulation time was investigated by the comparison of the flow field and pressure drop (see Fig.3 and Table 1).

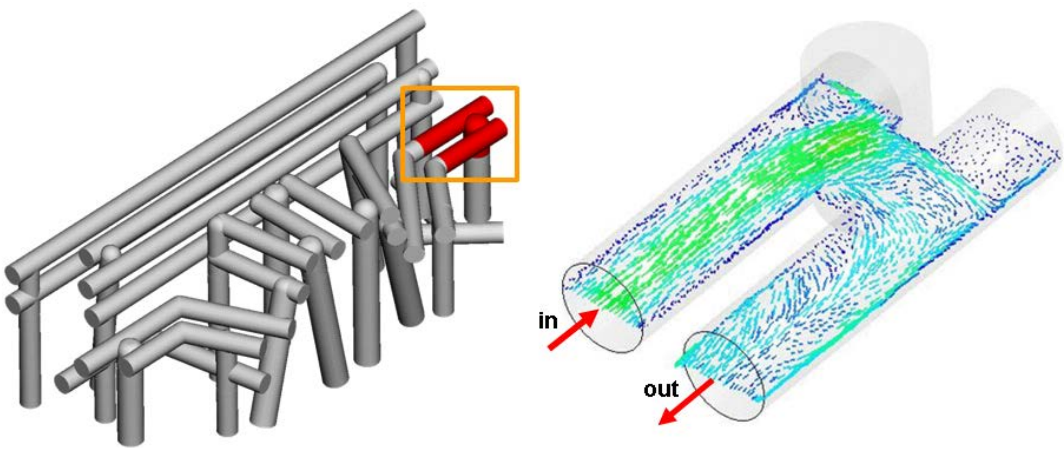

Fig.3: u-bend section of punch segment 1 (left) and vector plot of velocity (right)

el.len	#vol.elem.		dt	Δp		t_{Sim}	
[mm]	[ ]	[%]	[s]	[bar]	[%]	[h:m:s]	[%]
0.6	1 650 924	100	**0.001**	**0.248**	100	**3:56:48**	100
0.8	822 217	50	**0.001**	**0.248**	100	**1:35:40**	40
1.0	475 090	29	**0.001**	**0.240**	97	**0:37:27**	16
1.3	254 162	15	**0.001**	**0.229**	92	**0:14:22**	6
1.6	164 778	10	**0.001**	**0.232**	94	**0:07:48**	3

Table 1: pressure drop and simulation time at constant time step size by variation of mesh size

With this quantities, simulations of the flow through all of the four segments of the punch tool with all three turbulence models which are currently available in LS-DYNA's ICFD solver are performed and the results are compared. It turned out that a steady-state solution is reached after a physical time of about 0.5 s. Figure 4 shows a fringe plot of the surface pressure of the pipe of punch tool of segment 4 at a physical time of 0.4 s.

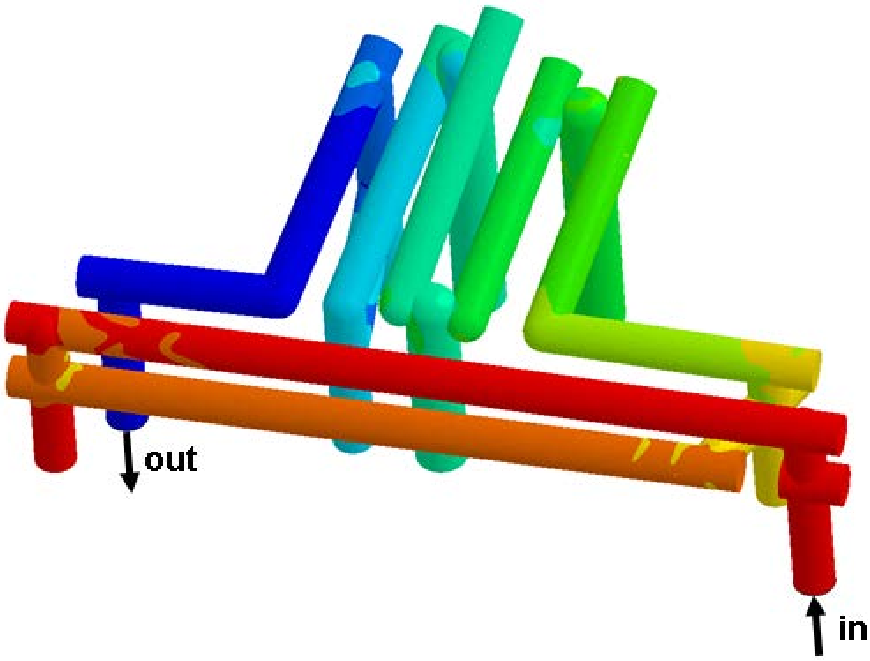

Fig.4: fringe plot of the surface pressure of the punch tool of segment 4

A comparison of the flow rate of simulations with the three different turbulence models VMA (Variational Multiscale Approach), k-epsilon and LES (Smagorinsky Large Eddy Simulation) give almost the same results for the simulations with VMA and k-epsilon model and slightly higher flow rates in the simulations with the LES model (Fig. 5), when the same mesh size is used in the simulation models of all three turbulence models.

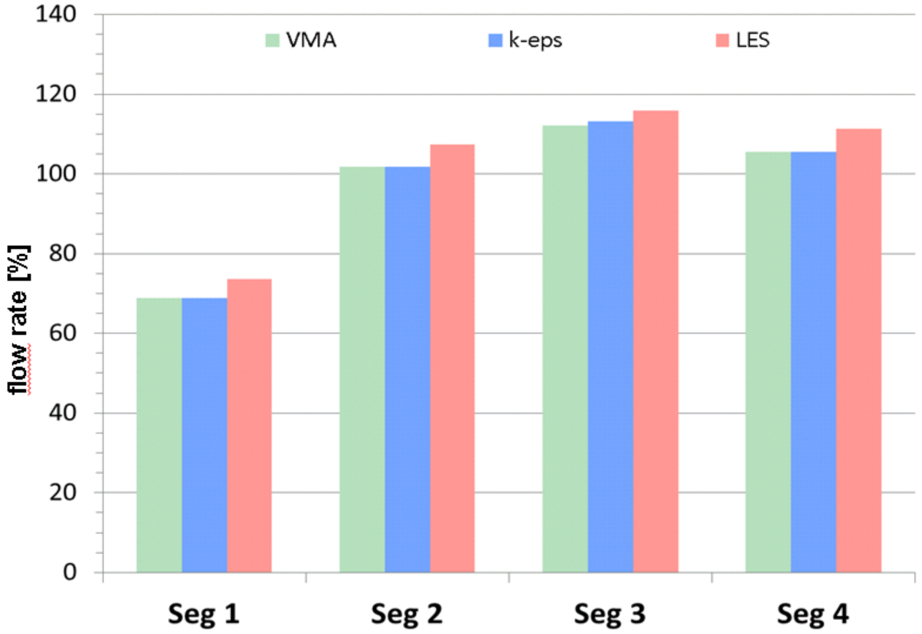

Fig.5: simulations with different turbulence models; comparison of flow rate (all segments)

When using the same mesh size and time step size for the simulations with the three different turbulence models, the comparison show longer runtimes with the k-epsilon model, which is a two-equation model, compared to the VMA and LES models (Fig. 6). At this point it has to be said that when using the LES turbulence model, usually a significantly finer mesh is needed compared to the other two turbulence models which would lead to considerable higher runtimes.

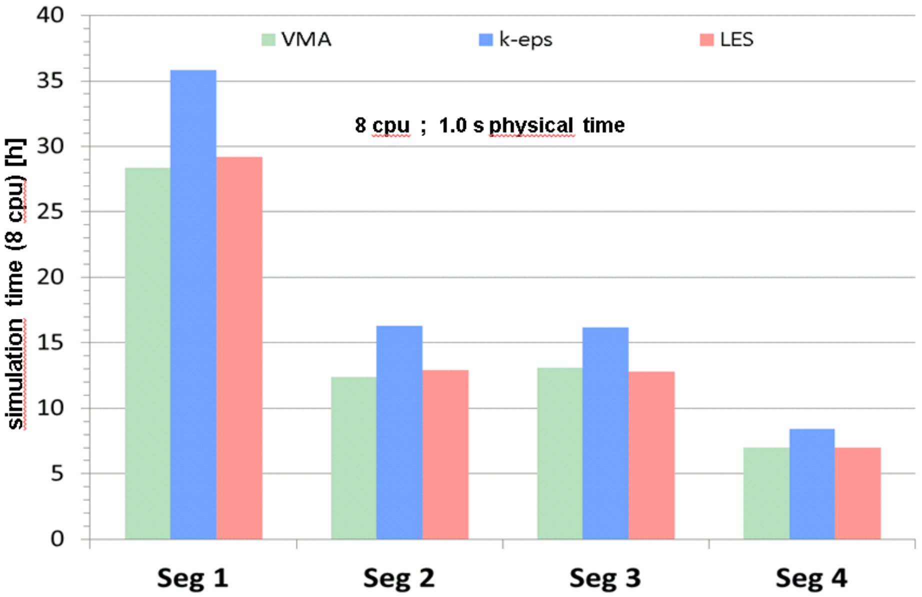

Fig.6: simulations with different turbulence models; comparison of runtimes (same mesh size

As a result of this investigation it was decided to take the VMA turbulence model because the results in terms of flow rate are the same as with the k-epsilon model but in a smaller runtime while the LES turbulence model was not a choice because of the higher requirements concerning the mesh size which leads to higher runtimes and memory usage compared to the other two turbulence models .

2.2 Thermal Coupled Simulation

For punch segment 4, a thermal coupled simulation of the solid tool (punch only) and the fluid domain was performed (conjugate heat transfer).

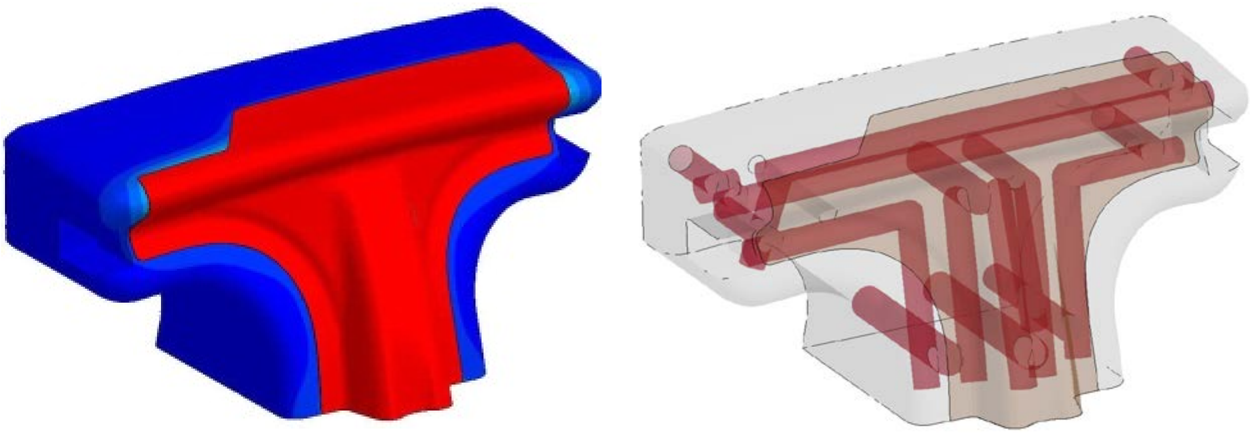

Fig.7: punch segment 4 with blank (red); left image shows a fringe plot of temperature at time t=0; right image shows the geometry of tool and blank in transparent mode

Since in LS-DYNA's ICFD solver the heat equation for the fluid is solved inherently, nothing has to be changed concerning the setup of the fluid problem. What has to be changed in the conjugate heat transfer simulations compared to the previously described CFD simulations is that a description of the solid tool has to be provided and that the ICFD solver has to be coupled with the thermal solver for solids. The coupling between these two solvers is a strong (monolithically) coupling. The intersection of the two domains with it's common boundary along the walls of the pipes is marked accordingly in the input. The meshes of this common boundary does not necessarily have to be coincident (Fig. 8).

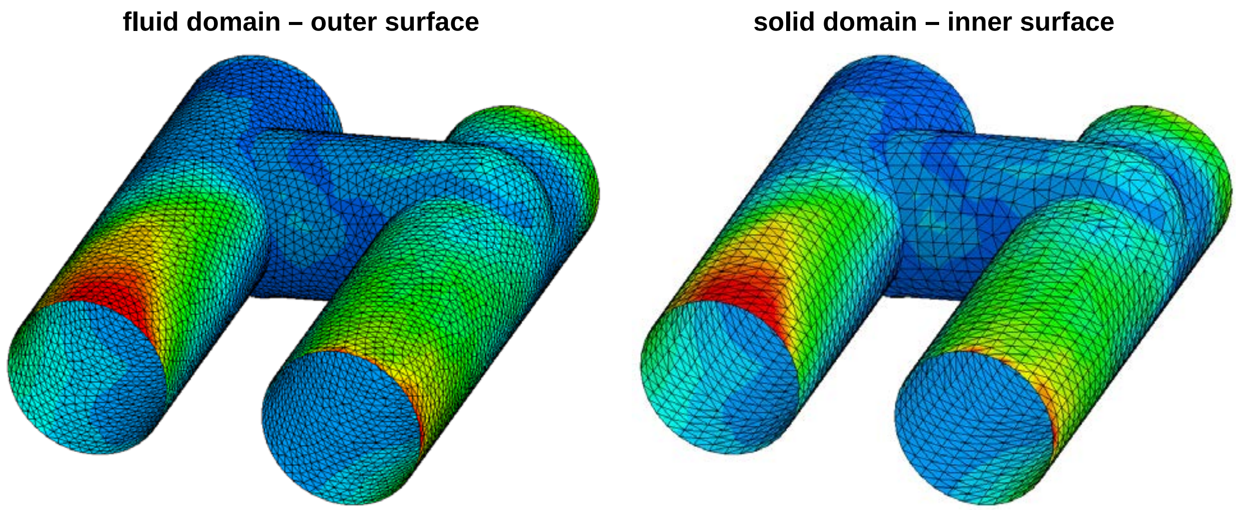

Fig.8: coupling: mesh and temperature distribution on the common boundary

The initial temperature distribution in the tool is determined beforehand using thermal-only simulations of multiple consecutive forming cycles where the temperature at the end of one cycle is used to initialize the tool temperature of the subsequent cycle. This temperature distribution in the tool is used as an initial boundary condition in the coupled simulation.

2.3 Discretization and Boundary Conditions

The fluid is assumed to have the fluid properties of water at a pressure of 1 bar. At the inlet, temperature and velocity, and at the outlet, pressure is defined to have a constant value while a non-slip boundary condition is assigned to the walls.

Tool and blank are defined as rigid bodies. Since the focus in this investigation was the heat transfer between tool and fluid, the forming of the blank was not part of the simulation. A completely shaped blank was used just from the beginning of the simulation instead. The thermal properties of the blank are those of boron steel 22MnB5 which is the solely used material in press hardening processes. The thermal properties of the tool are those of a standard material used for hot forming tools.

Shell elements are used for the discretization of the blank. The volume tool mesh consists of tetrahedron elements and is provided by the user. The volume fluid mesh is generated automatically by LS-DYNA's ICFD Solver integrated volume mesher. For the definition of the fluid domain, a surface mesh has to be provided. The resolution of the volume mesh in the boundary layer is controlled by the declaration of the number of refinement layers normal to the wall.

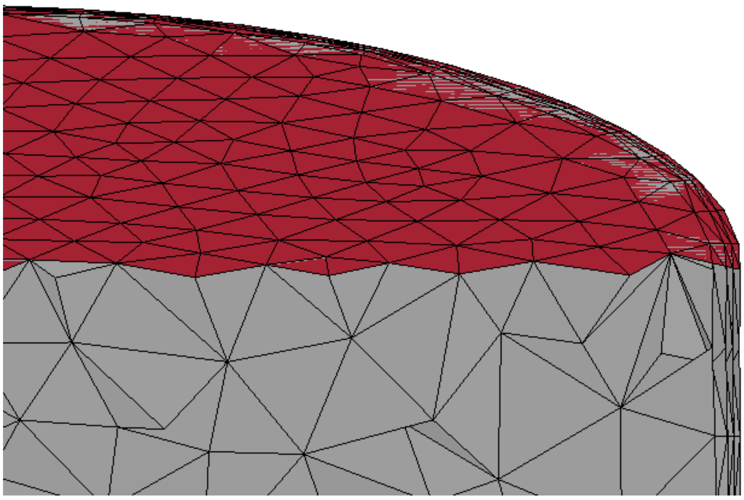

Fig.9: section of the automatically generated volume mesh of the fluid with boundary layer resolution

The heat transfer between blank and tool is defined by a thermal contact definition with a constant heat transfer coefficient *h=3000 W/(m²*K)* , which implies a constant contact pressure all over the contact surface. This is a simplified approach since the contact pressure and therefore the heat transfer coefficient in the real process varies significantly in different regions of the contact surface. This simplified approach was chosen since the focus in this work was not the heat transfer from blank to tool but the investigation of the coupling of ICFD and thermal solver.

The heat transfer coefficient between solid and fluid depends on the thermal properties of fluid and solid, the flow rate, the type of the flow (laminar or turbulent), the geometrical conditions, the surface roughness and the temperature range and is calculated inherently by the solver in the conjugate heat transfer simulation. The relationship between the heat flux q, the heat transfer coefficient *h* and the temperature difference ΔT is described according eq. (1)

$$h = \frac{\dot{q}}{\Delta T} \tag{1}$$

with

$$\Delta T = T_{\infty} - T_W \tag{2}$$

as the difference of the wall temperature T_W and the bulk flow temperature T_{∞}.

At the time when the simulations for the herein described investigation were performed, it was not possible to output or to look at the heat transfer coefficient along the coupling surface between fluid and solid.

The temperature difference ΔT of the wall temperature T_W and the bulk flow temperature T_∞ leads to a temperature profile in the near wall boundary layer of the fluid (Fig. 10).

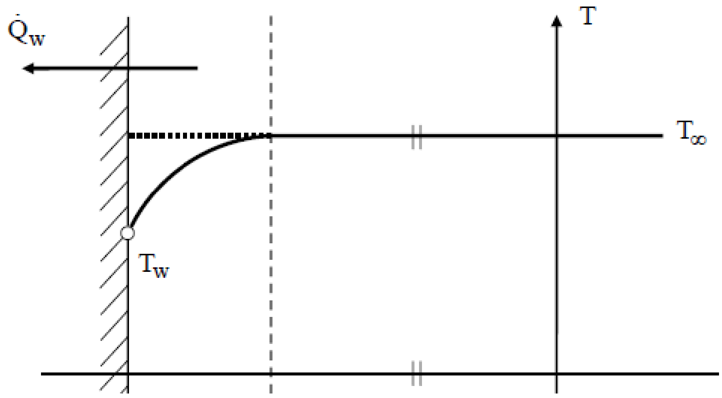

Fig.10: temperature distribution in the fluid near a solid wall

3 Results

Since there was no reliable experimental data available, it was not possible to compare the simulation results with experimental results. What was available were results from simulations of the pure CFD problem with results of simulations with other CFD codes. The comparison of the results of the pure CFD simulations with LS-DYNA's ICFD solver in terms of pressure drop and flow rate showed very good agreement with the results of the simulations with the other CFD codes.

The results of the coupled conjugate heat transfer simulations look feasible but couldn't be compared neither to experimental data nor to simulation results with other CFD codes.

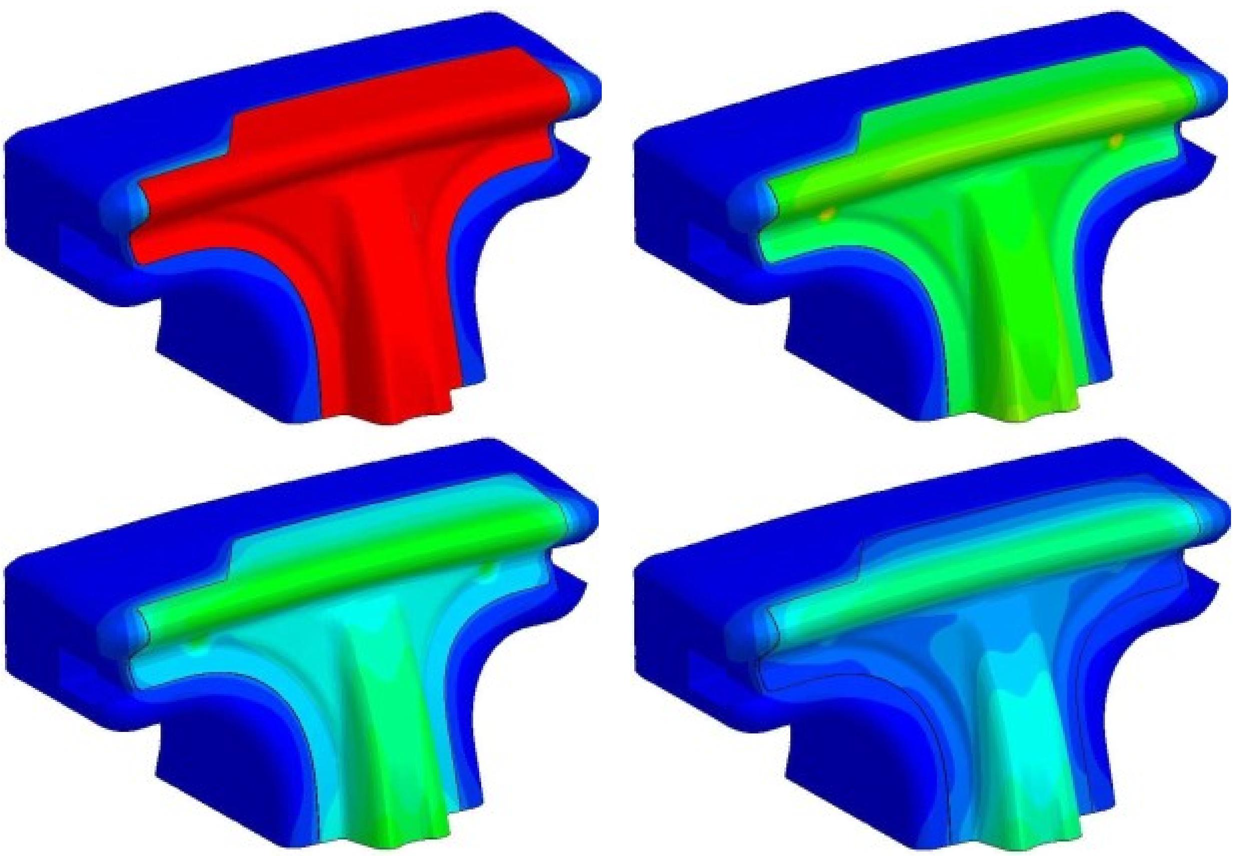

Fig.11: fringe plot of temperature in tool and blank during one cycle

4 Summary

The flow through the pipes of four separate segments of one of the two parts of a press hardening forming tool was simulated using LS-DYNA's ICFD solver. Only very few keywords were necessary to define the flow problem and the coupled conjugate heat transfer problem as well. Due to the automatic volume mesh generation facilities of LS-DYNA's ICFD solver combined with the fact that only very few parts are involved in the simulation, it is relatively easy to setup the simulation model. Users who come from structural analysis may be surprised in which short time it is possible to get results from the CFD simulation. However, some experience is necessary in the evaluation of the results.

The results of the pure CFD simulations showed good agreement with the results of simulations with other CFD codes. The VMA turbulence model turned out to be the best choice for this kind of problem.

The thermal coupling works well when some modelling rules like mesh sizes and time step size are met. It turned out that with increasing problem size there was an increasing chance to run into numerical instabilities for the coupled simulations.

There are limits concerning the coarsening of the fluid volume mesh in order to get physically reasonable results. Therefore, the runtimes are high and the memory usage for running the implicit simulation is considerable and is a limiting factor concerning the problem size. With the geometry which was used in the herein described investigation, it wouldn't had been possible to run all four cooling circuits in one simulation with the available hardware because of the tremendous memory consumption. Additionally, to set up a model of the real process, it would be necessary to include both tools, punch and die, into the simulation. Therefore, to run flow problems like the one presented in this paper, strategies have to followed to get the problem running with the available hardware devices and to get reasonable runtimes.

Since a steady state solution for the fluid flow is reached fast compared to the physical time of a complete forming cycle, it would make not much sense to run the complete forming cycle as a coupled problem. To run the conjugate heat transfer problem only until a steady state solution for the flow problem is reached and use the heat transfer coefficient determined in this coupled simulation as a boundary condition in a subsequent thermal only cooling simulation or a thermo-mechanically coupled simulation of a complete forming cycle without solving the fluid problem seems to be an applicable approach. So the following four-step approach could be a good procedure for this kind of process:

Step 1: Determine the heat transfer coefficient between sheet and tools in a thermo-mechanically coupled forming simulation with a constant temperature boundary condition along the pipe walls.

Step 2: Thermal only simulations of some consecutive forming cycles with a constant temperature boundary condition along the pipe walls to determine the temperature distribution in the tools.

Step 3: Coupled simulation (thermal + CFD) for a relatively short physical time until a steady state in the fluid is reached.

Step 4: The heat transfer coefficient between fluid and tool and the bulk temperature of the fluid along the pipe from this coupled simulation is used as a boundary condition in a

- 4a: thermal only cooling simulation or a
- 4b: thermo-mechanically coupled simulation of a complete forming cycle.

So the difference between Step 4b compared to Step 1 is that a prescribed temperature of the bulk flow T_∞ and the local heat transfer coefficient h is used as thermal boundary condition along the pipe walls instead the prescribed constant wall temperature T_W.

5 Literature

[1] Hochholdinger, B.: Simulation des Presshärteprozesses und Vorhersage der mechanischen Bauteileigenschaften nach dem Härten, Doktorarbeit, 2012

[2] Incropera, F.P., De Witt, D.P.: Fundamentals of Heat Transfer, 1981

[3] Caldichouri, I., Del Pin, F.: Incompressible CFD solver (ICFD) and FSI in LS-DYNA: Introduction and application, class notes, 2013

[4] Schade, H., Kunz, E.: Strömungslehre, 1989

[5] Çaldichoury, I.,Del Pin, F.: „LS-DYNA® R7: Conjugate heat transfer problems and coupling between the Incompressible CFD (ICFD) solver and the thermal solver, applications, results and examples", 9th European LS-DYNA Conference, 2013.
Name, Abbreviation of first name: "publication", issue, year, pages