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

Uli Göhner[1], Bruno Boll[1], Inaki Caldichouri[2], Tim Wicke[3]

[1] *DYNAmore GmbH, Stuttgart* [2] *LSTC* [3] *Volkswagen AG, Site Kassel*

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 achieved at the press hardening lines of Volkswagen 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 problem including 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.

Introduction

It is an essential requirement in the press hardening process, to get a particular amount of heat in a particular amount of time out of the blank. 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 the way to do this at the press hardening lines of Volkswagen is to run a fluid through them.

In order to reduce the manufacturing cost per part, it is desirable 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]. This means that there is a direct relationship between the efficiency of the cooling of the tools and the manufacturing costs per part.

Simulation Model

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 is an inverse approach because the pressure drop is a known and the flow velocity is one of the unknown quantities.

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.

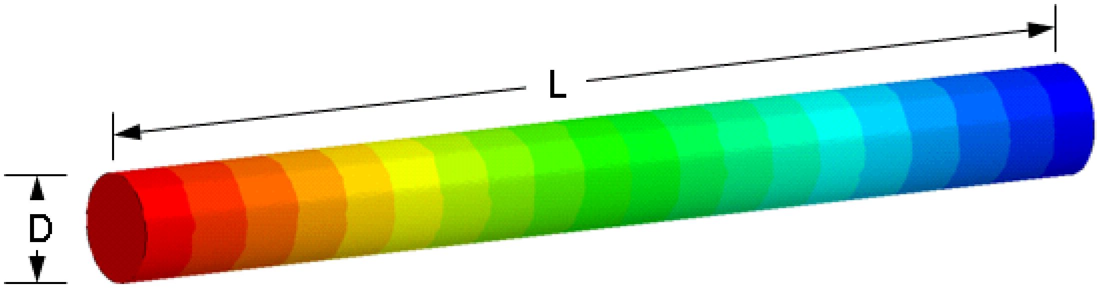

Figure 1: linear section of punch segment 1 with surface pressure (Poisseuille flow)

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 camparison of the flow field and pressure drop.

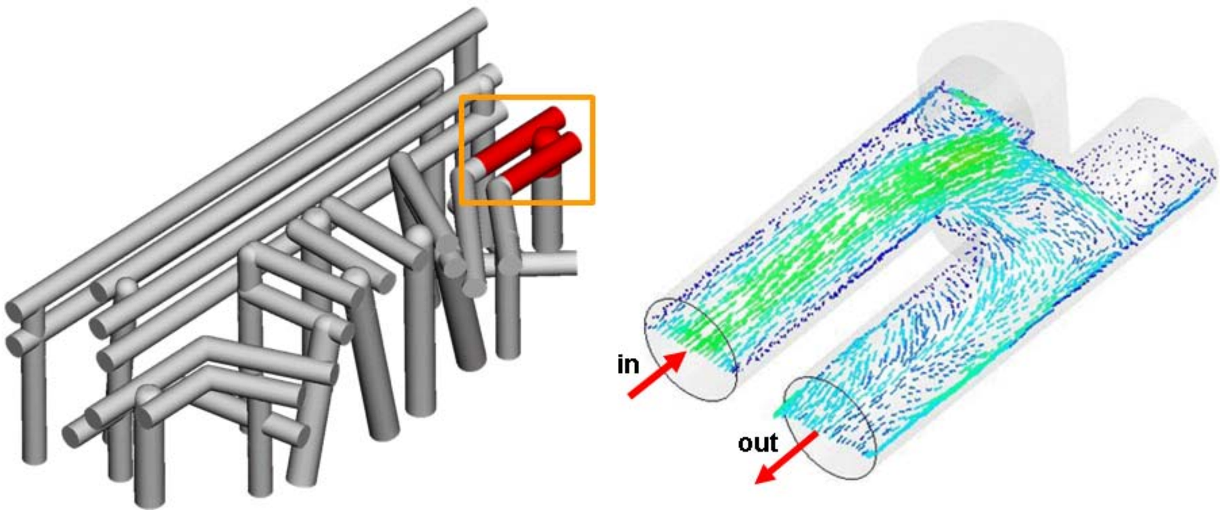

Figure 2: u-bend section of punch segment 1 (left) and vector plot of velocity (right)

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 these 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.

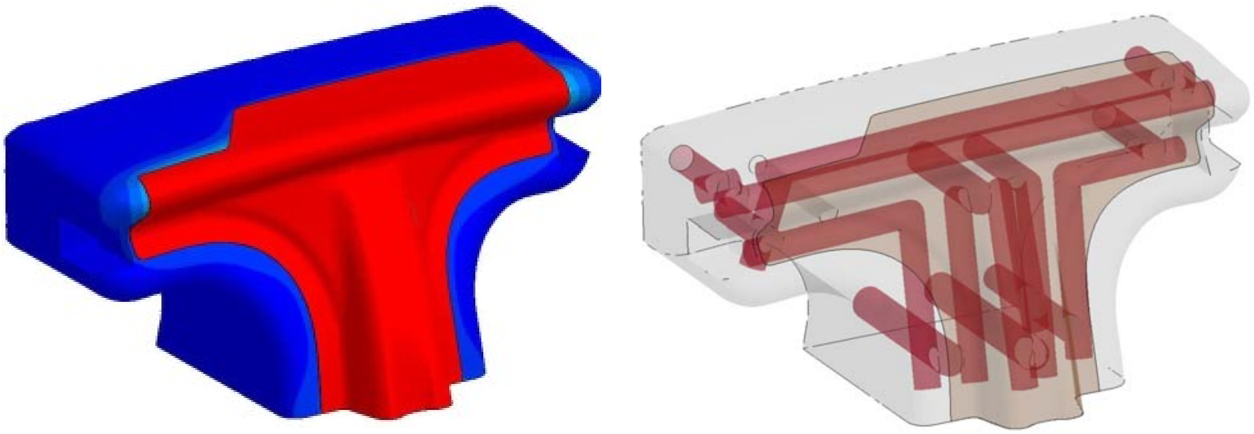

Figure 3: punch segment 4 with blank (red); fringe plot of temperature at t=0 (left); geometry of tool and blank in transparent mode (right)

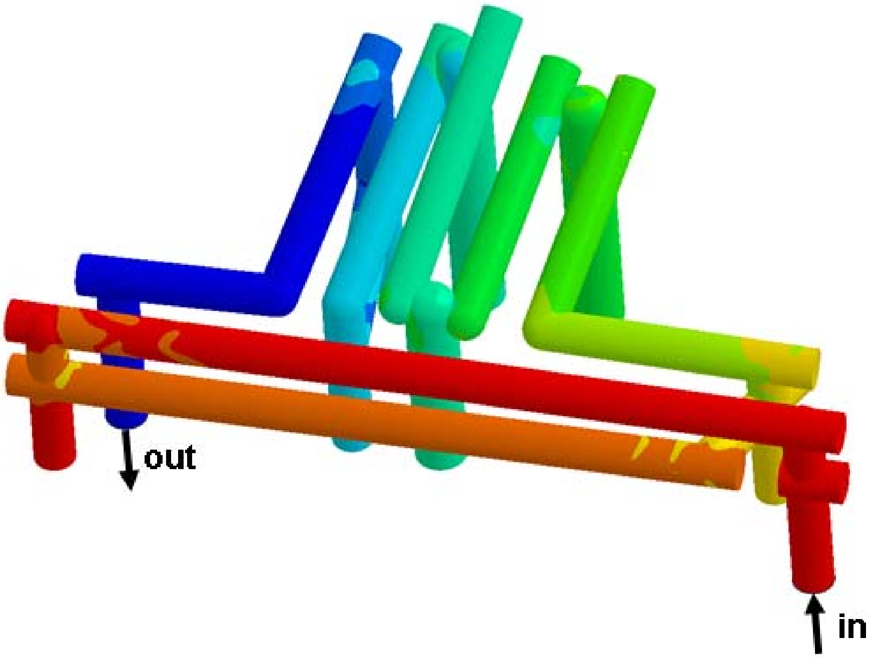

Figure 4: fringe plot of the surface pressure of the punch tool of segment 4

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) by using the same mesh size with all three turbulence models show good agreement (Figure 5).

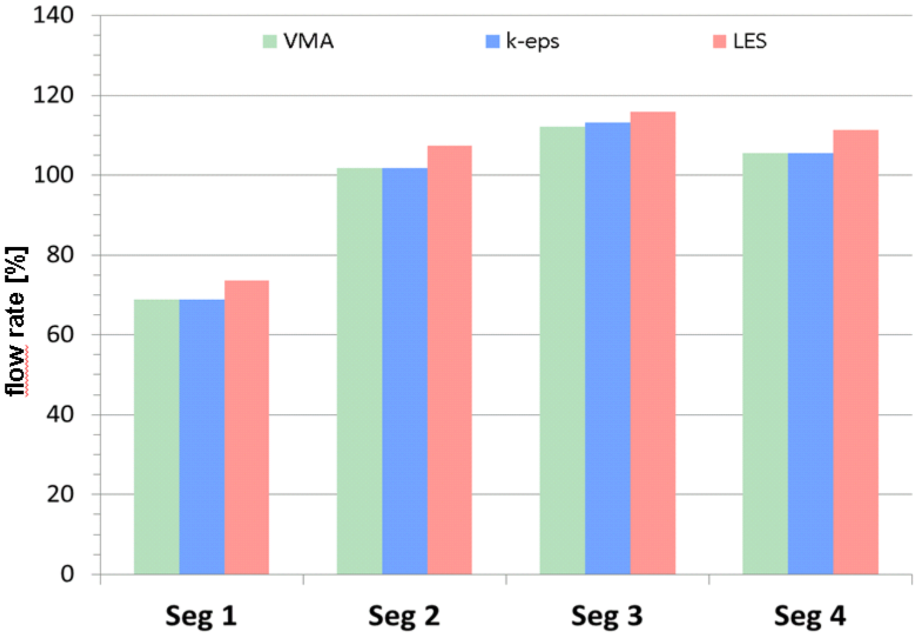

Figure 5: simulations with different turbulence models; comparison of flow rate (all segments)

Using the same mesh size and time step size, the comparison of simulations with the three different turbulence models show slightly longer runtimes with the k-epsilon model, which is a two-equation model, compared to the VMA and LES models. At this point it has to be said that when using the LES turbulence model, normally a significantly finer mesh is needed compared to the other two turbulence models which leads to considerable higher runtimes.

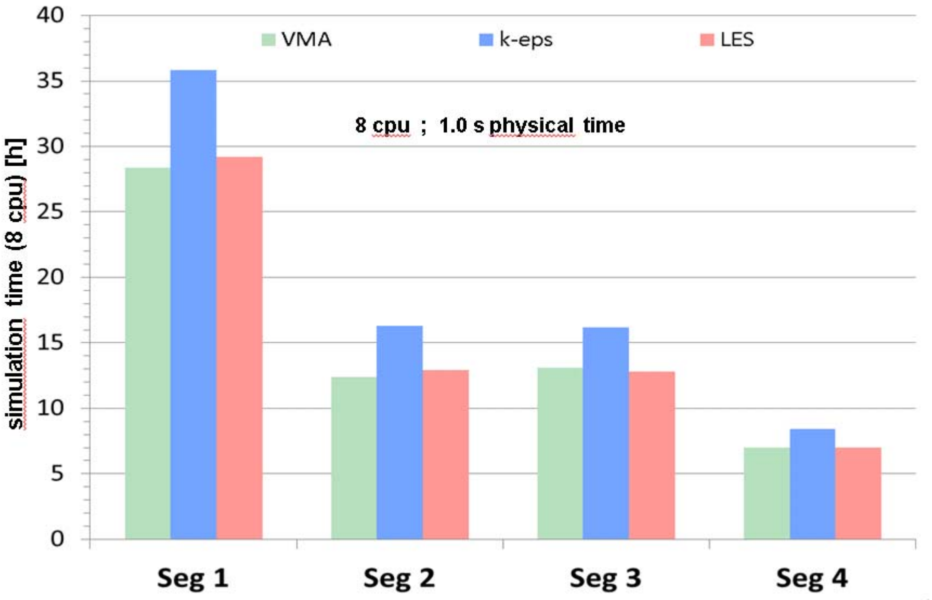

Figure 6: simulations with different turbulence models; comparison of runtimes (same mesh size)

Thermal Coupled Simulation

For segment 4, a thermal coupled simulation of the solid tool (punch only) and the fluid domain is performed (conjugate heat transfer). Since in LS-DYNA's ICFD solver the heat equation for the fluid is solved inherently, it 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 its common boundary along the walls of the pipes is marked accordingly in the input. The meshes of this common boundary don't have to be coincident (Figure 7).

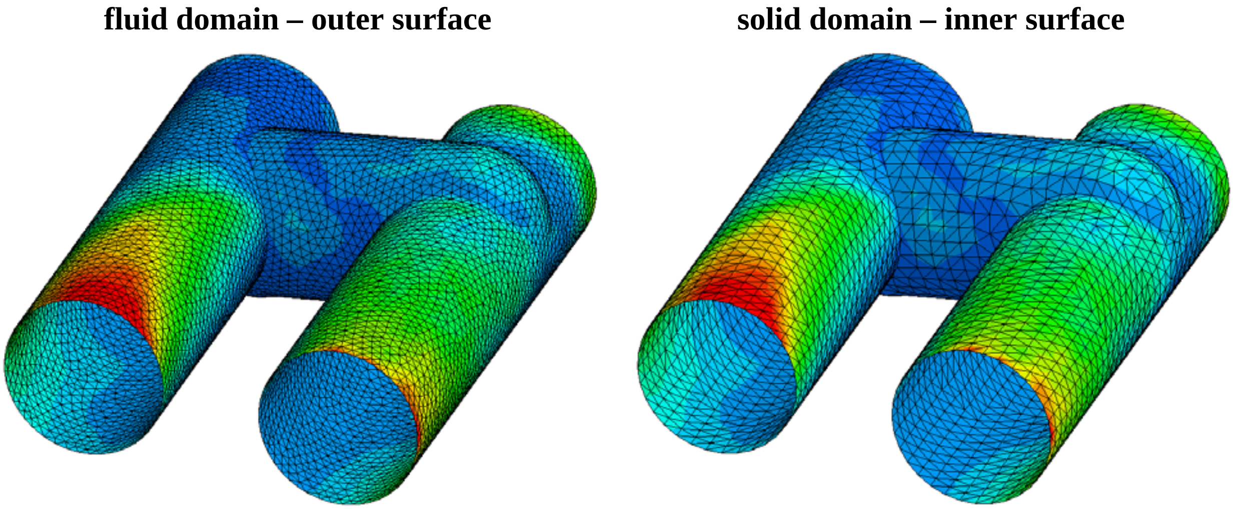

Figure 7: coupling: mesh and temperature distribution on the common boundary

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.

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 with a non-slip boundary condition for the walls.

Tool and blank are defined as rigid bodies. The forming of the blank is not part of this investigation. A completely shaped blank is used just from the beginning of the simulation instead. The thermal properties of blank and tool are those of the most usual materials used in press hardening processes.

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 element refinements normal to the wall.

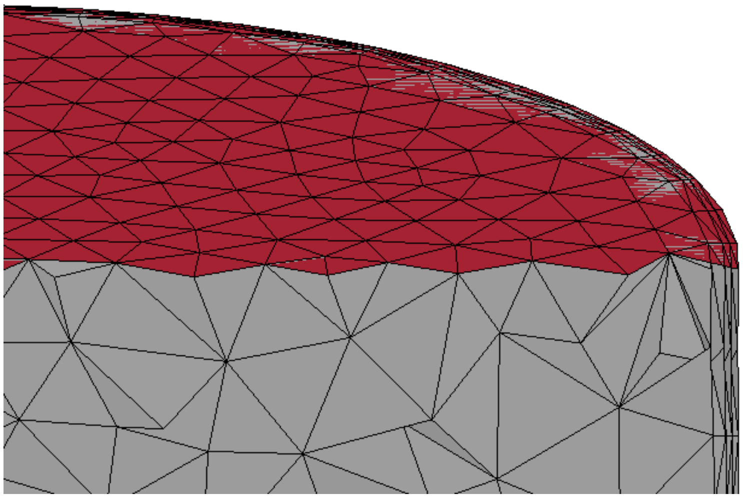

Figure 8: section of the automatically generated volume mesh with boundary layer resolution

The heat transfer between blank and tool is defined by a thermal contact definition with a constant heat transfer coefficient *h* all over the contact surface. This is a simplified approach since the heat transfer coefficient in the real process takes different values in different regions of the contact surface. This simplified approach was chosen since the focus in this work was the investigation of the coupling of ICFD and thermal solver.

The heat transfer coefficient between tool and fluid depends on the flow rate, the type of the flow (laminar or turbulent), the geometrical conditions, the surface roughness and the temperature range. It is not a value which is defined by the user like the heat transfer coefficient between blank and tool but is calculated by the solver in the conjugate heat transfer simulation.

Results

The thermal coupling works well when some modelling rules like mesh sizes and time step size are met. The bigger the problem the more sensible the simulation behaves. The results look feasible but couldn't be compared to experimental results. The runtimes and memory usage for the ICFD simulations are high and are a limiting factor concerning the problem size. To model the real process, it is necessary to include both tools, punch and die, into the simulation. Since a steady state solution for the fluid flow is reached fast compared to the cycle time, it would make not much sense to run a complete 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 simulation of the forming process without solving the fluid problem seems to be an applicable approach for this kind of problem.

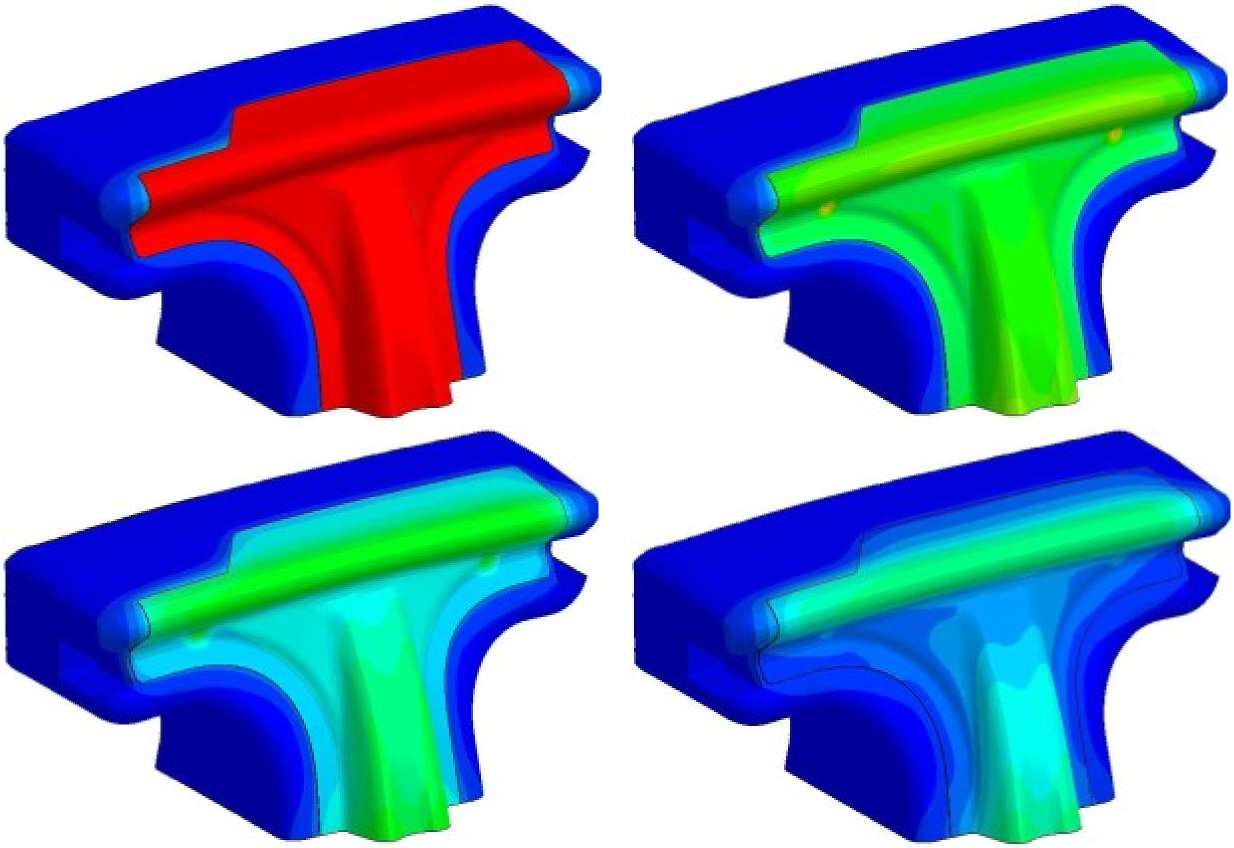

Figure 9: fringe plot of temperature in tool and blank during one cycle

References

[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.