A new approach for a thermo-mechanical coupled simulation of the hot stamping process

K. Kassem-Manthey*, Dr. H. Spiess†, K. Srivastava*, K. Wolf†

**GNS mbH, Am Gaußberg 2, 38114 Braunschweig, Germany*
†Fraunhofer SCAI, Schloss Birlinghoven, 53754 Sankt Augustin, Germany

Abstract: Nowadays, hot stamping of high strength steels is an attractive process for the manufacturing of crash relevant parts in the automotive industry. Due to forming at elevated temperatures, not only can the strength of the formed part be remarkably increased, but also the formability properties be significantly improved and the springback reduced to a minimum. In this paper, a new approach for the numerical simulation of the **hot stamping** process is presented. Based on the **M**esh-based **p**arallel **C**ode **C**oupling **I**nterface (**MpCCI**), a thermo-mechanical coupled simulation of an industrial part using two commercial codes is performed. On the one hand, the special-purpose forming simulation software **INDEED** is used to compute the mechanical (and coupled thermal) response, and on the other, the multi-purpose software **Abaqus** is used to compute the pure thermal response. Both solutions are incrementally coupled via **MpCCI**.

Keywords: Hot stamping, Thermo-mechanical coupled simulation, INDEED, MpCCI

1. Introduction

Hot stamping or press hardening is an innovative process for meeting the conflicting goal of manufacturing high and ultra-high strength but lightweight components in the automotive industry. This process attains a greater significance due to a limited cold formability of these materials and an increased expenditure in producing complex sheet metal components and higher operating forces needed for manufacturing high-strength parts. It involves the forming of a hot blank in the austenitic temperature range (900–950°C) together with simultaneous quenching (rapid cooling) in the die. The main advantages of this process are high strength (and hence better performance during crash), high formability and lower springback.

The hot forming process consists of the following sequence of operations: At first the blank is heated from room temperature to the austenization temperature (900-950°C) separately in an oven. The second step is to transfer the hot blank to the press chamber during which heat loss due to radiation occurs from the blank. Further heat transfer due to contact occurs when the blank is placed onto the die. The difference in temperature between the die and the blank leads to an inhomogeneous temperature distribution when they come into contact. This step is followed by the actual forming process which occurs very quickly in the manufacturing unit.

The final step is the cooling of the components involved, wherein, the rate of cooling plays an important role in determining the hardening properties of the deformed blank.

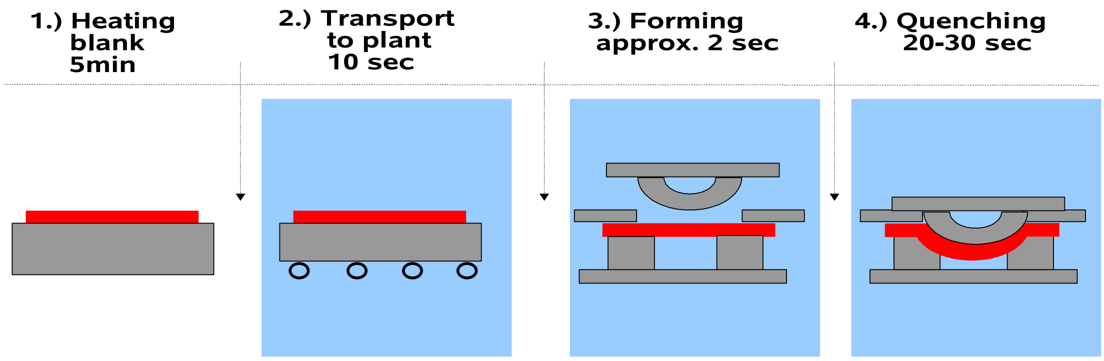

Figure 1: Stages of Hot Forming process

Numerical simulation of the hot forming process is a highly challenging task as it involves the consideration of the thermo-mechanical interactions that occur during the heating and the cooling process. The prediction of temperature distribution in the blank and the tools plays a very important role in this process. Apart from that, it requires a temperature-dependent hardening function to characterize the plastic deformation and also take into account the heat transfer between the blank and the tools and heat loss due to convection and radiation from the blank. The solid-solid phase transformation from Austenite to Martensite must also be considered in order to simulate the hot forming process in an accurate manner.

2. Multi-physics simulation by code-coupling

There is an increasing need for multi-physics simulations in various research and engineering fields. Fluid-structure interaction, magneto-hydro dynamics, thermal coupling, plasma computations or coupled manufacturing processes are only a subset of recent multi-physics activities. In most cases, a single (proprietary) simulation system cannot provide all necessary features, but the coupling of the best codes from each discipline enables more flexibility and simulation quality to the end user.

2.1 MpCCI interface concept

MpCCI (Mesh-based parallel Code Coupling Interface) has been developed at the Fraunhofer Institute SCAI in order to provide an application independent interface for the coupling of different simulation codes. MpCCI is a software environment which enables the exchange of data between the meshes of two or more simulation codes in the coupling region. Since the meshes belonging to different simulation codes are in general not compatible, MpCCI performs an interpolation between the nodes. MpCCI allows the exchange of nearly every kind of data between the coupled codes; e.g. energy and momentum sources, material properties, mesh definitions, or global quantities. The details of the data exchange are hidden behind the concise interface of MpCCI.

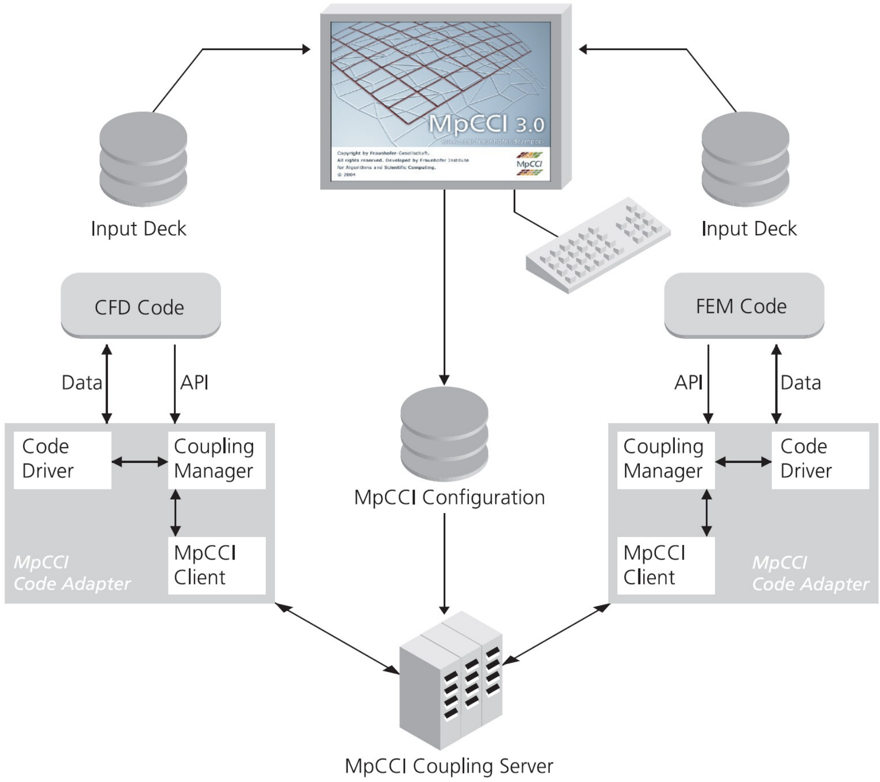

Figure 2: MpCCI code coupling concept

Within the MpCCI 3.0 system, the code adapters establish a direct connection between the MpCCI Coupling Server and the codes. These adapters make use of the APIs of the commercial codes and hence (in most cases) do not need modified versions of these codes. A code adapter is a library which will be linked to the code either statically or dynamically. Any code adapter consists of two modules - the Coupling Manager and the Code Driver. Additionally, the check of the model input data of each specific code, begin and termination of the codes is performed using shell scripts.

2.2 Supported Codes

The current version of MpCCI 3.0 supports Abaqus, Ansys, Fluent, Flux3D, ICEPAK, INDEED (prototype), MSC.Marc, Permas, StarCD, and RadTherm. Adapters for further codes like the 1D pipeline code Flowmaster are currently under development. Additionally, there is a strong demand for coupling certain dedicated in-house codes with commercial codes. Especially in the aerospace industry, engineers trust their own high-end in-house tools for aerodynamics simulations and prefer to use commercial FE codes for structural analysis. The universities and research institutes have developed their own codes, either for educational purposes or for evaluation of specific details in CFD or FEM.

2.3 Coupling quantities

The coupling requires an accurate update and exchange of different mechanical quantities involved in the simulation while taking into account the features of the coupled softwares. The basic quantity to be transferred is the mesh information i.e. nodal coordinates and connectivities of the elements. Also, information related to the mechanical variables like strains, stresses and forces needs to be accurately communicated between the coupled softwares.

3. Numerical simulation of hot stamping process

As already mentioned, numerical simulation of the hot stamping process is highly complex where, the mechanical deformation due to temperature changes during the forming process must also be considered. In this paper, a procedure is outlined for coupling the Abaqus thermal interface with INDEED's metal forming interface in order to simulate the hot forming process and thus taking into account the coupled thermo-mechanical response.

3.1 Forming simulation with INDEED

INDEED (INnovative DEEp Drawing) is a commercial implicit Finite Element Method based software especially tailored to meet the requirements of the metal forming industry. It can be used to simulate a wide range of metal forming processes like tube bending, roll forming, flanging, press joining, crashforming, hydro-forming and hydromechanical deep drawing. INDEED has highly specialized element formulations (e.g. solid shell elements with double-sided contact and linearly distributed transverse normal stresses), contact algorithms and material models allowing for different combinations of hardening models like isotropic, kinematic and distortion hardening, for a precise prediction of the plastic strains and stresses. These material models are especially suited to capture the high degree of nonlinear effects that occur during the forming process. Operations like cutting and trimming can also be easily simulated as INDEED allows for the modelling of the complete sequence of operations involved in the physical production process.

3.2 Heat transfer analysis with Abaqus

Abaqus/Standard can be used to perform heat transfer analysis for problems involving conduction, forced convection and boundary radiation. The temperature field can be calculated without any knowledge of stress/deformation in an uncoupled heat transfer analysis. Heat transfer problems can be classified either as steady-state or transient**,** linear or nonlinear. Besides these, the thermal interactions like gap radiation, conductance and heat generation between contact interfaces can be included. Abaqus has a special purpose element library for heat transfer simulations. For simulating the hot forming process, an uncoupled heat transfer simulation will be performed with Abaqus and the coupling effects will be taken into account in INDEED.

3.3 Coupling concept

By means of coupling INDEED with Abaqus via MpCCI, it is possible to account for both heat distribution in the blank as well as the tools. In a sheet metal forming simulation, the tools are normally assumed to be rigid bodies. This means that only the surfaces of the tools which might come into contact with the blank need to be discretized. If the heat distribution within the tools is to be accounted for, they have to be modelled with solid elements. This results in quite large FE models since the contact surfaces of the tools require extremely fine meshes to account for curvature. For this reason, it will be assumed here that the tool is at a constant temperature whose value should be properly chosen in order to take into account the heat transfer between the tool and the blank in an acceptable manner.

The different stages and their treatment in the numerical simulation of a hot stamping process are as follows: The state of the blank after heating is modelled to be having an initial temperature of 950°C in Abaqus. The transfer of the blank to the die, which takes about 6-8 seconds, is a coupled thermo-mechanical process. The cooling of the blank due to radiation and convection can be simulated by Abaqus. However, for a feasibility analysis and to save computation time, this step can be neglected with an assumption that the blank cools down by about 50°C during the transfer process. Hence the initial temperature of blank is taken to be at 900°C. In reality too, radiation effects are only significant at temperatures over 600 °C but are still small compared to convection and conduction and hence is neglected throughout the process.

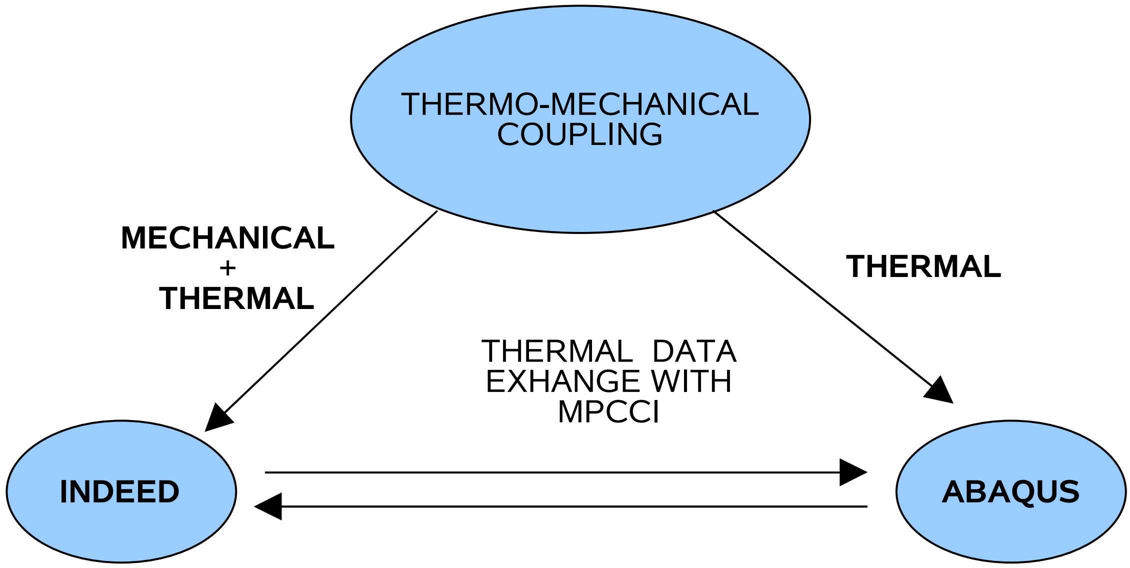

Figure 3: MPCCI Coupling between Abaqus and INDEED for hot forming

At every increment of the forming process, the temperature at the nodes of the blank are computed using Abaqus through the MpCCI interface. For this, the contact configuration of the blank, the die and the tools is required (i.e. information about nodes and surfaces in contact) and this is computed using INDEED. This is a purely mechanical step. Using this information, the uncoupled thermal step is performed using Abaqus in which, the temperature distribution in the blank and the tools arising due to contact between the tools and the blank is computed. Heat conduction across the contact surface between blank and die is calculated by means of the equation $q = \kappa\,(\theta_A - \theta_B)$, where q is the heat flux per unit area crossing the contact surface from point A on the die to point B on the blank, with θ_A and θ_B being the respective temperatures at these points. κ is the so-called gap conductance and depends on the contact pressure and on temperature. For a precise evaluation of the heat transfer between the blank and the die, the contact pressure as well as the size of the contact surface has to be carefully calculated within

INDEED. This contact leads to an inhomogeneous heat distribution in the tools and the blank. The thermal parameters influencing this step are the heat transfer coefficient, heat capacity of the materials and temperature distribution within the blank and tools. The temperature values at nodes are communicated to INDEED with the MpCCI interface wherein, the corresponding temperature values at the Gauss points are obtained using the same shape-functions that are used to discretize the geometry (iso-parametric formulation). The strains arising due to this "thermal loading" are computed and added to the mechanical strains.

To account for temperature-dependent material behavior the elasto-plastic material models in INDEED have been modified in such a way that the elastic constants are functions of the temperature and an arbitrary number of hardening curves can be defined for different temperature values. The hardening curve corresponding to other temperatures is then obtained by interpolating between the hardening curves defined for the material. However, this does not alter the procedure for calculating the plastic response during the forming process. The friction coefficient also needs to be defined appropriately. The stresses and the press forces occurring due to the deformation are computed by INDEED and hence the thermo-mechanical interactions are accounted for.

After forming the blank, the quenching process of the blank also needs to be modelled. As the temperature of the tool is assumed to be constant, and the heat transfer rate between tool and blank is proportional to the temperature gradient, this assumption leads to a higher rate of cooling as in reality the temperature of the tool is increasing. However, for a feasibility study during the product manufacturing cycle, this assumption leads to quick and acceptable results.

The advantage of using a coupling of this form is to utilize the features of two softwares specializing in their own domains in the most extensive manner. The schema for this coupling is shown in Figure 3.

4. Summary

A new concept of coupling two finite element codes (INDEED and Abaqus) specializing in their own respective domains for performing hot forming simulations has been proposed. The coupling of the respective codes is performed using the MpCCI software environment developed at Fraunhofer institute. An uncoupled thermal simulation step is performed in Abaqus and the temperature values at nodes are subsequently transferred to INDEED and considered in a temperature dependent hardening model. The thermo-mechanical coupling effects are computed in INDEED and so are the thermo-mechanical interactions accounted for.

Literature

[1] Prof. Dr. Ing Ralf, Kolleck: Neuere Entwicklungen in der Werkzeug und Umformtechnik. Einführung in den Maschinenbau, TU Graz, Oktober 2006

[2] Prof. Dr.-Ing. habil. Kurt Steinhoff: Moderne Thermomechanische Prozessstrategien in der Stahlumformung, Stahl-Akademie,Veranstaltung 13/07, Mai 2007, Düsseldorf

[3] Prof. B.-A.. Behrens, P.Olle : Numerische Simulation des Presshärtens unter Berücksichtigung der Gefügeumwandlung, 6. LS-Dyna Forum 2007, Frankenthal

[4] Abaqus Thermal Analysis Manual