A coupled flow, heat and structural analysis tool to reduce creep during heat treatment of titanium components

Ulrich Heck, Martin Becker, Ralf Paßmann, DHCAE Tools, Krefeld

Volker Hardenacke, Otto Fuchs KG, Meinerzhagen

1 Summary

Titanium alloys represent an essential pillar of the lightweight construction concept of modern aircraft. They offer excellent specific strength and at the same time high corrosion and temperature resistance. However, the manufacturing processes require optimization, as high material requirements from the forging process and subsequent heat treatment may lead to material removal of 80% to 90% by machining. For economic and ecological reasons, the goal of optimized manufacturing is to reduce the distortion of the component.

Creep during component cooling in quiescent air or by forced convection in a circulating air-cooling chamber has been identified as a significant distortion parameter after heat treatment. In order to understand the processes during cooling, it is necessary to model the entire process. The next step is to optimize the process. One part of the model is based on computational fluid dynamics (CFD) with heat transport: The open-source CFD toolbox OpenFOAM was adapted to take into account all the main cooling processes in an efficient way. These major heat transfer mechanisms include in particular the dominant radiation heat exchange of the component with the support grid in the initial phase, combined with energy release through microstructure transformation. Later in the cooling process, convection through the ambient air and heat conduction in the solid bodies become more important. In a second step, different coupling methods between the flow, heat transfer and the structure have to be verified in order to implement the creep analysis. However, it turned out that by calculating the temperature field on the CFD side in a conjugate heat transport analysis, the problem did not require an iterative coupling between CFD and structural analysis, and only the time-dependent temperature field from the CFD had to be transferred to the structural solver Abaqus for the creep analysis. On the structural mechanics side, extensions via the programming interfaces were implemented to ensure efficient processing of the large amount of data due to the time-dependent temperature fields for all nodes. A graphical interface was also integrated into Abaqus to allow the user to easily generate the necessary data for the exchange.

An intensive validation phase took place for the CFD through temperature measurements and for the structural analysis through deformation measurements. For this purpose, the measurements were carried out on different components and under different cooling conditions. Finally, the model was integrated into an automated process. The whole process was also optimized to reduce computation time and to achieve stable computations. This process is now being used in further development to optimize distortion.

2 CFD modelling

The model for the flow analysis consists of a titanium component mounted on supports. The supports in turn are mounted on a grating. Cooling in quiescent air and in forced convection is considered. A conjugate heat CFD analysis is carried out with the CFD toolbox OpenFOAM. The actual component as well as the supports and the grate are considered as solids (see Figure 1).

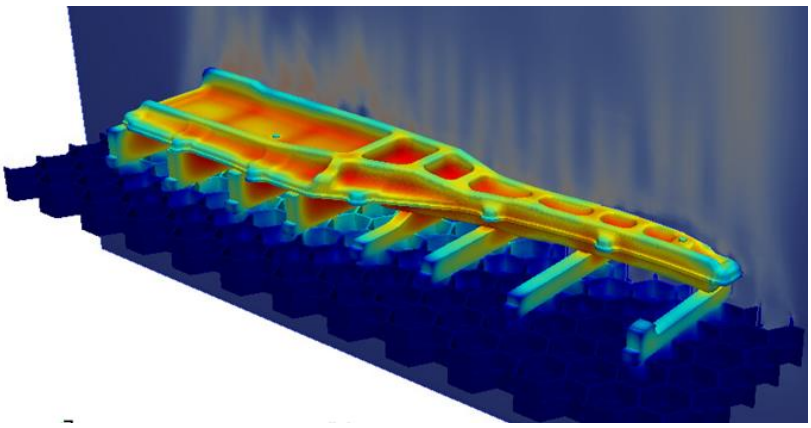

Figure 1: CHT model in CFD analysis

The analysis is transient with a RANS turbulence modelling (k-omega SST). Natural convection, radiation (modelled with an fvDOM approach) and heat conduction are to be considered as heat transport mechanisms, whereby a structural transformation also takes place in the titanium component, during which energy is also released. The energy release due to the microstructural transformation is converted into temperature-dependent cp values by a simplified model. Depending on the cooling conditions, the following approaches are used:

2.1 Cooling in quiescent air

In the case of cooling in quiescent air, in reality the component is moved out of the furnace on a grate and cooled in the factory hall. One difficulty in CFD modelling here is the time scale on which the transient cooling must be considered: typically, cooling times of about 15 minutes have to be taken into account, whereas a stable time step in the simulation is usually a few milliseconds. Adaptive time step sizes and optimizations in the solution procedures are used here in particular. For the development and optimization of the procedures, measurements were carried out on various components at different positions in order to validate the calculation procedures.
Figure 2 shows the normalized temperature curve and the cooling rate of some comparison positions. In general, a good agreement between simulation and experiment is achieved. The plot of the cooling rates shows that the initial phase of the cooling process is characterized by a release of energy during the microstructure transformation.

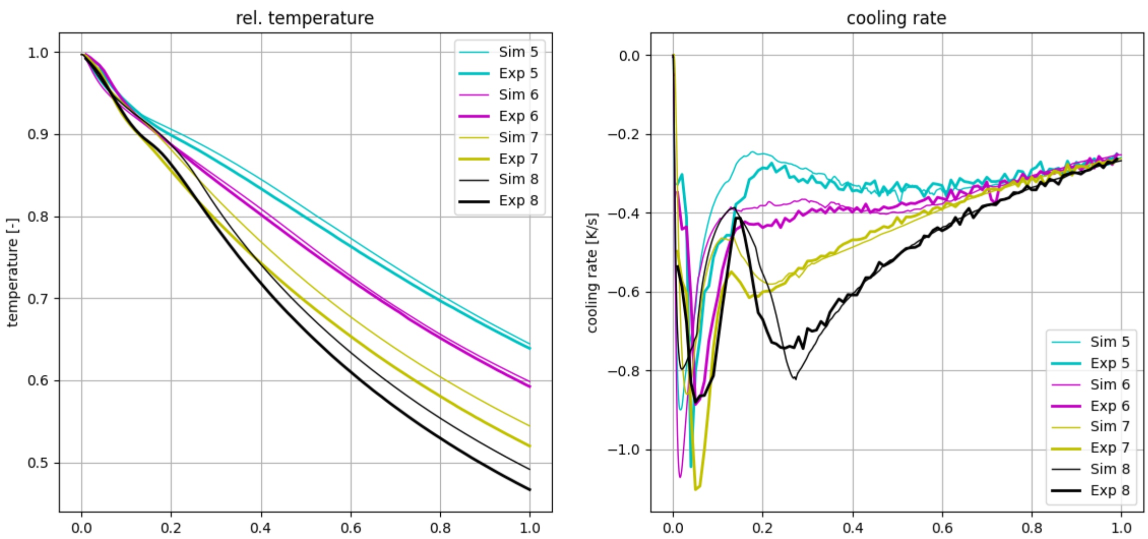

Figure 2: Temperature and cooling rate vs. time for cooling in quiescent air

2.2 Cooling in rapid air-cooling chamber

For cooling in the rapid air-cooling chamber, the chamber with the individual nozzle fields and the grate supports must also be modelled in the geometry. The mechanisms of heat transport in rapid air-cooling chamber differ significantly from cooling in quiescent air: While cooling in quiescent air is dominated by radiation, cooling in the rapid air-cooling chamber is dominated by forced convection due to the exhausted air from the nozzle field. This requires a different strategy for modelling the heat transport. Since the gas velocity from the nozzle field is constant, a so-called frozen flow approach can be used here. In this case, the flow field is calculated in a stationary manner and in the subsequent transient simulation only the energy equation with radiation, conduction, structural transformation, and convection is solved in the constant velocity field. Here, however, different phases must be taken into account in the boundary conditions, such as the transport phase in which the component is transported from the oven into the cooling chamber. Figure 3 also shows a comparison between simulation and experiment for some measuring positions. The agreement is satisfactory, but not quite as good as for cooling in quiescent air. The positioning of the nozzle field was identified as a possible cause. A detailed analysis of the temperature data from the flow analysis showed that a temperature difference of 50 K can occur between a stagnation point under a nozzle and the intermediate position between two nozzles. In the comparison between simulation and experiment, this remains as an uncertainty, since here no agreement is possible in the positioning of the component between simulation and experiment.

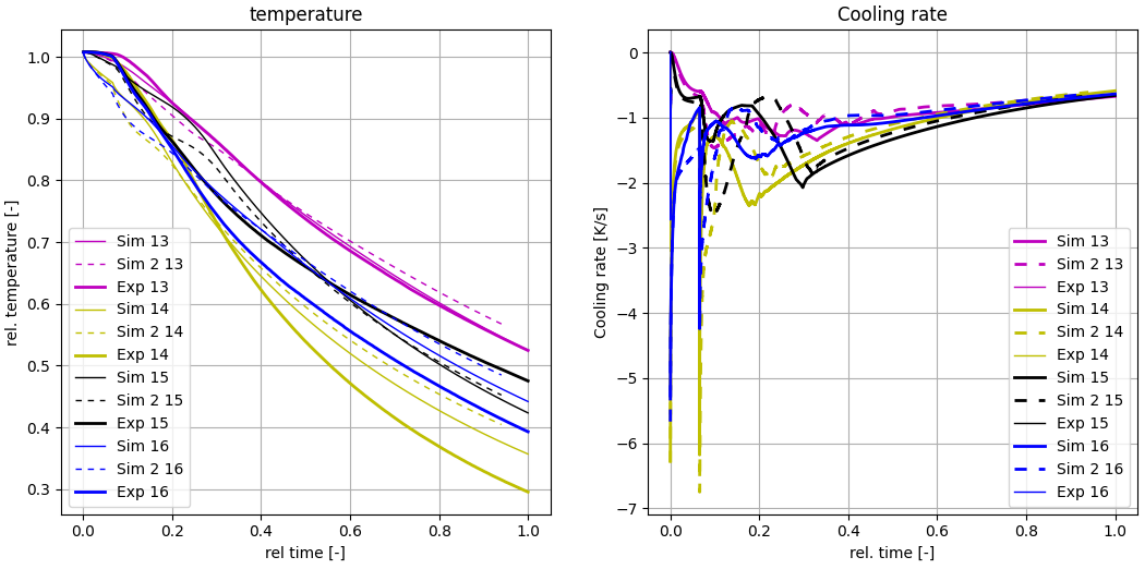

Figure 3: Temperature and cooling rate vs. time for cooling chamber

3 Coupling CFD and structural analysis

Creep has been identified as a major cause of deformation of titanium components during heat treatment. Therefore, it is necessary to perform a structural-mechanical calculation in addition to the flow simulation. For the coupling, different approaches were investigated:

On the one hand, a bi-directional coupling between the conjugate heat transfer CFD and the strength analysis was considered. A bi-directional coupling would be necessary if the temperature field in the titanium component calculated in the CFD analysis is influenced by a component deformation. This would be conceivable, for example, if the component detaches from some supports due to deformation: The heat conduction between the supports and the component calculated in the CFD would be influenced by the gap. However, numerical tests of the bidirectional coupling showed that the heat conduction across the supports is of minor importance and thus a simple unidirectional coupling can be used. Thus, the two problems of flow with heat transport in the solid on the one hand and deformation due to a determined temperature field on the other hand could be solved independently. This considerably reduces the numerical effort required to carry out the simulation.

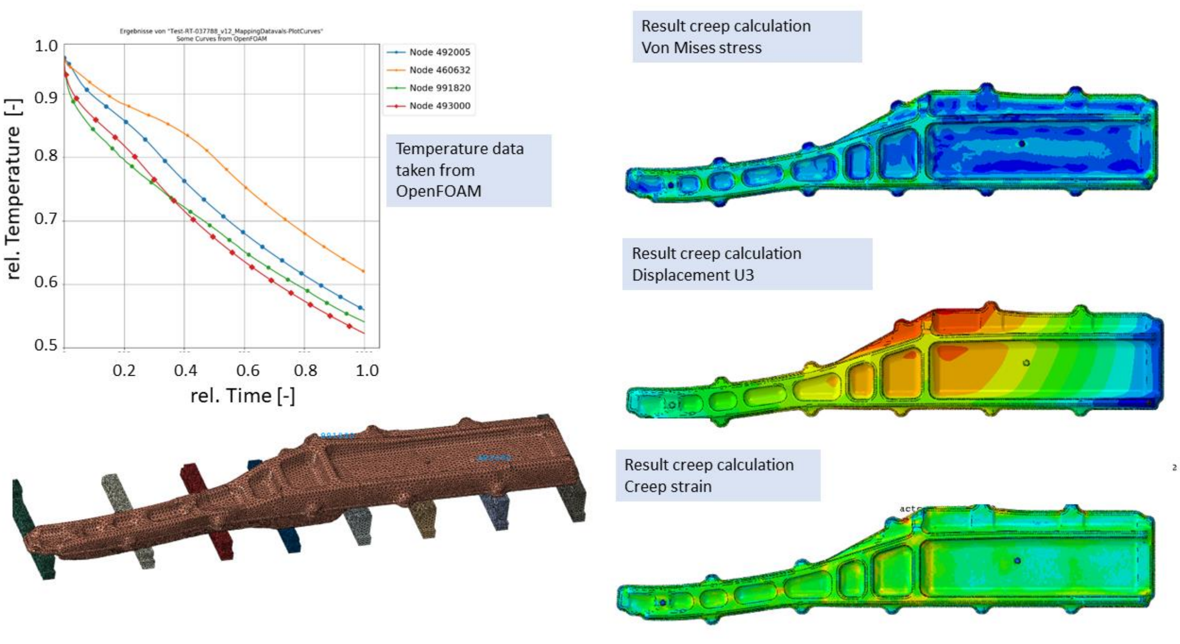

Figure 4: Structural results from mapped temperature fields

To perform the unidirectional calculation, a mapping of the transient temperature field is carried out (shown in Figure 4). The coupling realized between OpenFOAM and Abaqus as a structural solver. With the sampling function in OpenFOAM, the temperatures at the respective times are mapped to the separately exported nodes of the FE mesh. Via the Fortran interfaces provided by Abaqus, the data are processed, and the temperatures are used as loads for the thermal strain and creep modelling in the structural analysis. In this case, only the titanium component needs to be modelled. The supports can be modelled as rigid bodies in contact in the titanium component.

4 Automation

The CFD workflow is fully automated. The geometry exported in STL format for grate, supports and component serves as input. With a parameter file for the individual boundary conditions, the case is automatically meshed via Python and shell scripting, provided with boundary conditions and finally calculated. For the meshing, the OpenFOAM-internal mesh generator snappyHexMesh is used, which generates a hexahedron-dominant polyhedral mesh. The meshing and simulation are carried out on a server under Linux. A monitoring tool allows the simulation progress to be checked from a Windows desktop. Finally, the mapping of the temperature data to Abaqus is also done via the monitoring tool.
The structural-mechanical simulation with Abaqus runs preferably GUI-based within a Windows environment at the end user. The interfaces provided by Abaqus CAE GUI input fields for model preparation (such as gap treatment), automated processing of the temperature fields and the definition of the individual steps for the different heating and cooling phases were created.

5 Application

The overall model is used in development to optimize the cooling conditions. It was shown that the creep of the component can be significantly reduced if there is no considerable temperature difference between the top and bottom of the component. Cooling in the previous process cools the underside more slowly because the grate underneath retains heat longer. In the simulation, additional reflectors can simply be installed for testing purpose, for example see Figure 5. It can then be analysed for the specific components whether cooling takes place both uniformly and at the required cooling rates.

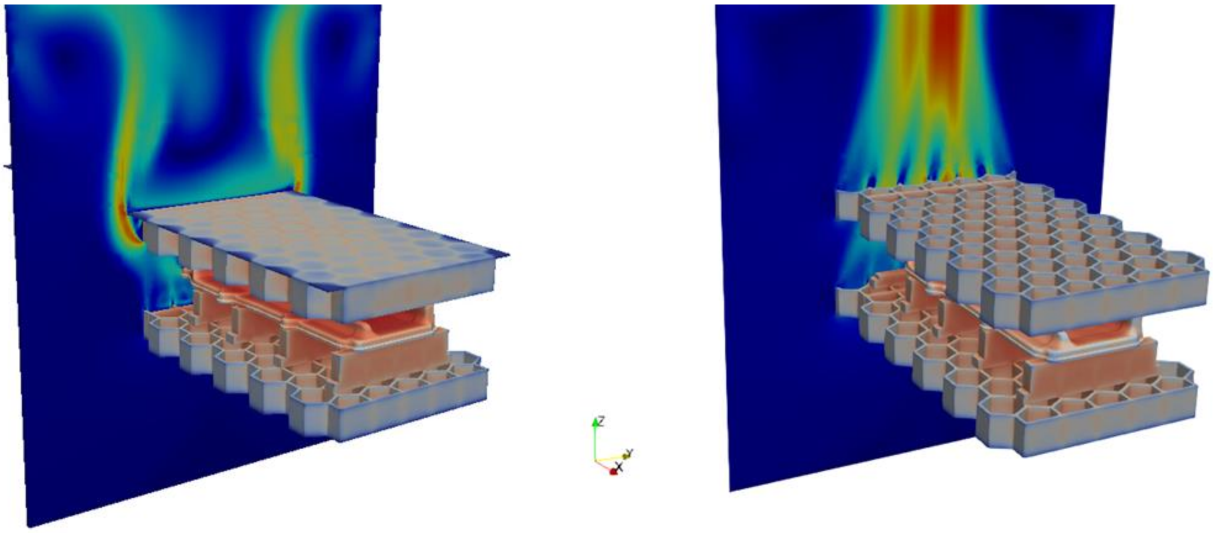

Figure 5: Heat transfer optimization with different reflector designs

6 Acknowledgment

The project was supported by the Federal Ministry for Economic Affairs and Climate Action on basis of a decision by the German Bundestag.