Fast Calculation Methods for the Modelling of Transient Temperature Fields in a Steam Turbine during Pre-Warming Operation

Piotr Łuczyński, David Erdmann, Dennis Toebben, Mathias Diefenthal, Manfred Wirsum, Institute for Power Plant Technology, Steam and Gas Turbines, RWTH Aachen University, 52056 Aachen, Germany
Klaus Helbig, GE Power AG, 68309 Mannheim, Germany
Wolfgang Mohr, GE Power AG, 5401 Baden, Switzerland

ISROMAC
Since 1985

Long Abstract

Introduction

In turbomachinery design the accurate prediction of the lifetime consumption is one of the most challenging issues. Nowadays, due to the thermodynamic optimization of operating parameters and the increasing requirements for flexibility of turbomachinery applications new engineering solutions are needed. Changes have to be introduced not only to the design but also to the operating strategy of turbomachines. For the reason of improving the accuracy of the life cycle prediction it is essential to consider temperature gradients (thermal loads) within the components causing thermo-mechanical fatigue. In order to account for the thermally induced stresses in the rotor and casing of turbines fast methods are required to predict metal temperatures as a part of the standard design process. A coupled transient simulation of fluid and solid states results in very high calculation times, as the timescales of fluid and solid differ by magnitudes. With regards to investigations of heat transfer phenomena within the boundary layer it is crucial for the numerical approach to calculate the fluid flow in these regions as accurate as possible.
In the present work different calculation methods are investigated for pre-warming of a high-pressure (HP) / intermediate–pressure (IP) steam turbine with hot air. The concept of turbine pre-warming with hot air has been introduced by General Electric [1] and serves to prevent negative impact on turbine components due high temperature gradients. The further development of pre-warming strategies requires a comprehensive understanding of fluid-solid states interaction. Thus, a new simulation approach, based on the modification of the material properties of the solid state is presented and compared to various calculation methods described in literature. In addition, the new approach is compared to an experimentally validated method which has been developed in an earlier research project [2].

Methods

Basically, two calculation methods of transient fluid/solid heat transfer can be distinguished. In the coupled method or the conjugate heat transfer method (CHT), a CFD calculation is iteratively coupled with a conductive FEA calculation. The continuity of the temperatures and the heat fluxes at the fluid/solid interfaces is ensured by solving the energy equation at the coupled boundaries. Examples of this method can be found in Bohn et al. [3], [4].
The uncoupled method is based on the assumption of constant heat transfer coefficients, implying a linear relation between the convective heat fluxes across the boundary layer and the driving temperature differences. Based on this assumption the heat transfer coefficients are determined by CFD Simulations and used as boundary conditions for subsequent FEA calculations. Examples of this approach are the heat transfer calculations of rotor and stator cavities published by Alizadeh et al. [5] and Lewis et al. [6]. In comparison to the coupled method, the uncoupled method can save calculation time, since no iterative solution is necessary at the fluid/solid boundary. A time accurate calculation on basis of the coupled method requires an extreme computational effort, since the time

scales of the convective and the conductive heat transfer differ by magnitudes (cf. He and Oldfield [7]). Simplifying assumptions have to be made to reach acceptable calculation times for a coupled method simulation of a transient heating process as a consequence (Figure 1).

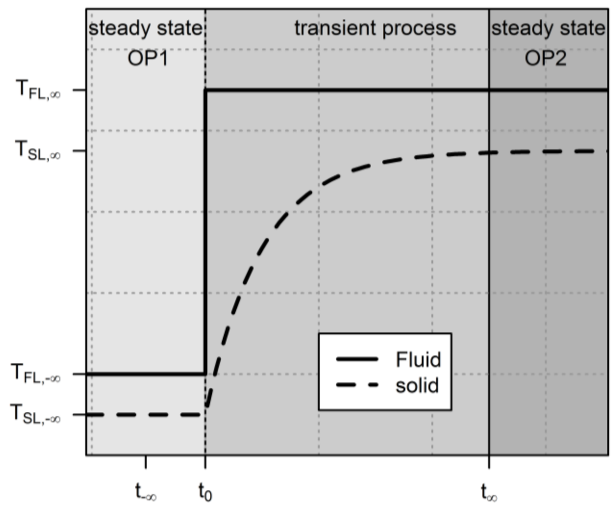

Figure 1. Qualitatively idealized behavior of fluid and solid temperatures in a thermal shock test [2]

In the present paper three numerical methods are presented to calculate transient fluid/solid heat transfer in a thermal shock process of a steam turbine. In a first step extensive simulations are conducted, in which the fluid and the solid calculations are coupled over the whole transient process. This method employs the coupled method as described before (called Frozen Flow method (FF)). The fluid equations are partly frozen respectively not solved over certain periods of time. A detailed description of this method is given in Diefenthal et al. [2]. The FF solution serves as a reference for other simulation approaches because of experimental validation.

As a further approach, an uncoupled method for the calculation of the transient temperature fields has been implemented in the numerical model. This method is able to capture the exponential behavior of the heat transfer coefficients in a transient process (called transient finite element analysis method-exponential (TFEA-EXPO)).

In addition a third coupled method is presented. In this approach the material properties of the solid state are modified in order to shorten the duration of the heating process. With this modification the timescales of fluid and solid are equalized resulting in significantly reduced calculation times. Thus, the approach (called Equalized Timescales method (ET)). With this approach no simplifications of the fluid state are required and fluid and solid state are calculated coupled over the whole transient process.

Finally, all the presented simulation approaches are compared with regard to calculation time and solution accuracy.

References

[1] K. Helbig, C. Kuehne, and W. F. D. Mohr, "A warming arrangement for a steam turbine in a power plant", EP2738360A1, June 4, 2014

[2] M., Diefenthal, C., Rakut, H., Tadesse, M., Wirsum, T., Heuer, „Temperature Gradients in a Radial Turbine in Steady State and Transient Operation”, International gas turbine conference, Tokyo, Japan, 2015

[3] D., Bohn, J., Ren, K., Kusterer, “Conjugate Heat Transfer Analysis for Film Cooling Configurations with Different Hole Geometries,” ASME, Paper No. GT2003-38369, 2003

[4] D., Bohn, T., Heuer, K., Kusterer, „Conjugate Flow and Heat Transfer Investigation of a turbo charger”, ASME Journal of Engineering for Gas Turbines and Power 127(3), pp. 663-669, 2005

[5] S., Alizadeh, K., Saunders, L., V., Lewis and J., Provins, “The Use of CFD to Generate Heat Transfer Boundary Conditions for a Rotor-Stator Cavity in a Compressor Drum Thermal Model,” ASME Turbo Expo GT2007-28333, Montreal, Canada, 2007

[6] L., V., Lewis, J., I., Provins, “A non-Coupled CFD-FE Procedure to Evaluate Windage and Heat Transfer in Rotor-Stator Cavities,” ASME Turbo Expo GT2004- 53246, Vienna, Austria, 2004

[7] L., He, M., L., G., Oldfield, “Unsteady Conjugate Heat Transfer Modeling,” ASME Journal of Turbomachinery, 133, No. 3, July, 2011