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

Piotr Łuczyński[1]*, David Erdmann[1], Dennis Többen[1], Mathias Diefenthal[1], Manfred Wirsum[1], Klaus Helbig[2], Wolfgang Mohr[3], Stefan Reitschmidt[3]

ISROMAC 2017

International Symposium on Transport Phenomena and Dynamics of Rotating Machinery

Hawaii, Maui

December 16-21 2017

Abstract

In turbomachinery design the accurate prediction of the life cycle is one of the most challenging issues. Nowadays, due to the thermodynamic optimization of operating parameters and the increasing requirements for flexibility of turbomachine applications new engineering solutions are needed. These compulsory changes must be introduced not only to the design but also to the operating strategy of turbomachines. For the accuracy improvement of the life cycle prediction it is essential to consider the temperature gradients (thermal loads) within the components which cause thermo-mechanical fatigue. In order to account for the thermally induced stresses in the turbine wheel and housing 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 regard 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 accurately as possible.

In the present work three 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 invented by General Electric [1] and serves to prevent a negative impact on turbine components due the 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. Additionally, some changes to a validated numerical model of a turbocharger are applied to extensive calculation method, resulting in additional calculation time reduction.

Keywords

transient simulations — conjugate heat transfer — pre-warming — thermal shock — steam turbines

[1] *Institute for Power Plant Technology, Steam and Gas Turbines, RWTH Aachen University, Aachen, Germany*
[2] *GE Power AG, Mannheim, Germany*
[3] *GE Power AG, Baden, Switzerland*
***Corresponding author**: luczynski@ikdg.rwth-aachen.de

INTRODUCTION

The situation on world energy markets is changing dynamically. The rising awareness of the environment and especially of the reality and severity of climate change has marked the beginning of a real breakthrough in modern concepts of energy production. Instead of striving for the cheapest possible solutions, societies value environmental-friendly, energy-sufficient and sustainable energy generation. Electricity production in big, centralized units is reduced, and increased, locally with wind, solar power and other renewables. In the European energy market, the share of renewable energies is increasing considerably over the last years. Regarding Germany, the politics support a priority feed-in electricity tariffs to encourage the use and development of new energy technologies. Similar policy mechanisms may be encountered in other European countries like France, Denmark or Poland. For almost all renewable energy resources the key issue is variability and a certain unpredictability. Therefore, wind parks or solar power always need back-up energy sources. Although 24-hour weather forecasts are reasonably accurate, wind speeds or radiation intensity can change quickly and unpredictably so that the back-up capacity should have the ability to react as quickly as possible. Fluctuation only in wind power may have a strong influence on balancing electricity supply with demand. An example is the sudden reduction of electricity generation from wind mills in Texas on February 26, 2008 [2]. Due to rapid decrease of wind speed the generated wind power dropped from 1.7 GW to only 700 MW. In combination with evening peak in power demand of about 3 GW, the grid operator was forced to use a rapid demand reduction of 1.1 GW for sake of system stability [2]. This has been reflected in electricity prices peaks and it offered additional profit to the electricity producers, who were able to cover the unbalanced demand.

In the light of the above considerations, the current situation changes the role of the conventional power plants on the energy market and produces new, challenging requirements for modern steam turbines regarding the flexibility and increasing number of startups due to the high share of fluctuating power input of renewable generation. Especially hard coal fired power plants already have to shut down on the weekends, for periods of up to 60 hours [3]. As a consequence a cold start-up is required afterwards. The frequent start-up and shut-down processes increase the lifetime consumption of the turbine components as they are not designed for this operational pattern. Furthermore, the conventional plants have to start-up as fast as possible in order to guarantee the security of supply and grid stability.

Fast start-up causes high metal temperature gradients. Hence, with decreasing start-up time the thermal stress, particularly of the thick-walled components, increases. This leads to an overly high lifetime consumption. Especially critical are the HP and IP steam turbines due to the high temperature difference between inlet steam and rotor temperature after a long period of idleness. Thermal stress is proportional to the difference between rotor surface and rotor average temperature. Faster loading means higher stress, which is limited not to overuse lifetime. Cold starts have the highest temperature difference from start to base load, thus, a start-up usually requires five to eight hours, which does not longer correspond to recent market requirements [1].

For the sake of shorter start-up procedure, the crucial components of steam turbine have to either be kept warm during the idle times or efficiently pre-warmed before operation. Considering a conventional power plant it is commonly known, that the thick-walled components staying in contact to the hot steam limit the start-up speed [1]. On the side of the boiler the temperature gradients in the heat exchanger headers, valves and drums have to be minimalized during start-up procedure. On the side of the steam turbine the special treatment requires the rotors and split lines. The pre-warming or warm-keeping procedure reduces the temperature difference between the steel and the steam and hence, the evoked stresses within the component decrease.

Recent studies have shown different possibilities to heat-up the rotor and the casing. In [4], three different methods to increase the temperature of the steam turbine in a solar power plant have been investigated: heat blankets on the surface of the housing, gland steam temperature increase and additional insulation. These investigations were conducted on the basis of a simplified calculation model. In [5], the work by analyzing three further methods for heating up the turbine with steam has been continued: Gland steam pressure increase, back pressure increase and barring speed increase. Due to its simplicity and possibility of application for different types of turbines the idea of heat blankets attached to the outer surface of the housing gained the interest not only of the university researchers but also of big-league turbomachine producers, like Siemens or GE. To the best of the author's knowledge, both companies have not decided yet to present the final results of their investigation to the international scientific community. Nevertheless it should be mentioned that for two-casing turbine the effectivity of rotor pre-warming may be limited by air heat transfer resistance. Additional problem for the heat blankets method is the pre-warming of internal turbine components like valves.

The mentioned methods based on pre-warming with hot working medium very often have to be analyzed under the consideration of windage effects. Windage occurs generally at low volume flow and high rotational speed and causes a temperature increase due to friction and dissipation, as found in literature [6], [7], [8]. These findings imply that it is possible to heat-up a turbine by windage. A further approach for heating up or keeping warm the turbine with hot air was developed [9]. With an additional hot air warming cycle, warm or hot start-up conditions can be provided even after long idle times. Depending on the expected idle time, hot air is used to keep the HP / IP steam turbine warm or to pre-warm it. This air is heated up by means of an electrical heater and passed with a fan through the turbine, while the rotor is continuously rotating. This approach serves as a basis for the present investigations. Based on the single stage turbine model the heat transfer phenomena in warm-keeping operation have been analyzed in [10]. In the next step the pre-warming operation should be investigated. For this purpose, the additional research into the area of numerical simulations and time-efficient calculation approaches has to be investigated.

For the optimization of the startup behavior, calculations of the time-depending temperature distribution within the thick-walled components are required. Due to extremely high computational times necessary for transient *Conjugate Heat Transfer* (CHT) simulations the fast calculation approaches have to be developed. The numerical methods allowing to calculate heat exchange and transient temperature fields within the turbine in pre-warming operation with regard to thermomechanical fatigue as well as their comparison are described in the following sections.

1. NUMERICAL MODEL

On the basis of the literature research and in cooperation with the industrial partner General Electric, a numerical model of a single stage turbine is developed to calculate the heat transfer in a steam turbine during pre-warming operation. It is based on the numerical model created for the warm-keeping investigation, which was discussed in [10]. Therefore, in the following sections only the most relevant information concerning model setup as well as the changes introduced to original settings are presented.

The numerical model aims to analyze the convective heat transfer and to develop different calculation approaches. As the highest steam temperatures and thermally induced stresses arise particularly within the first steam turbine stage, the heat transfer investigations focus on the first stage. In this

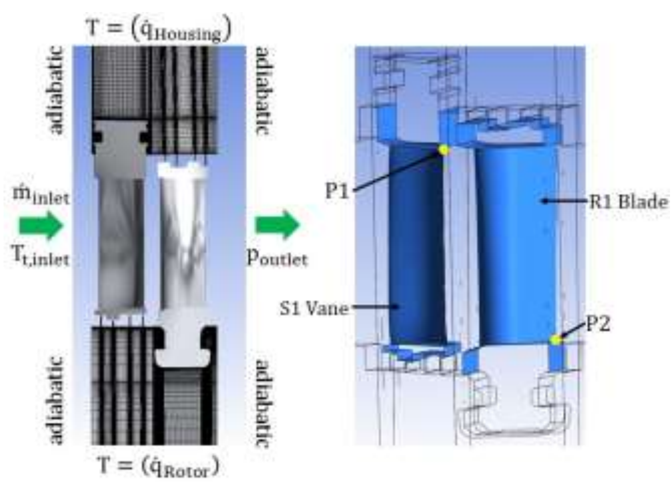


Figure 1. Numerical model of the solid body with boundary conditions and surface denotation

model, the flow and the heat transfer in the solid state are assumed to be of periodic nature in the circumferential direction. With this assumption, a single blade passage (approx. 2.5°) of the turbine wheel is extracted and meshed. All domains included in the numerical model are discretized by hexahedral cells, which are conformed to each other at the interfaces between fluid state and solid body in the turbine stator and rotor. In the fluid boundary layer a dimensionless wall distance lower than y+ = 1.0 is implemented. In order to exactly calculate the heat transfer, the temperature distribution and the flow field in the boundary layer of the solid body has a high resolution as well. The mesh study conducted in [10] has proven, that spatial discretization has a negligible influence on values of averaged Reynolds- and Nusselt-Number.

The fluid state is assumed to be a fully turbulent ideal gas (air) with constant fluid properties. To model the thermal boundary, the Low Reynolds kω-SST turbulence model is used. Constant material properties are applied for the solid state as well.

In general, concerning time dependency of the flow, two types of calculations are required to numerically depict pre-warming process: steady and transient simulations. The main goal of steady state simulations is to generate the boundary conditions for investigated transient calculation approaches. The setup of steady state, as well as of transient models is discussed in the following sections.

In Figure 1, the numerical model with its boundary conditions and the used denotations is shown. As presented, the axial walls of the model are considered to be adiabatic. In the present work, the pre-warming procedure has been modelled as thermal shock process, in which the turbine inlet temperature $T_{T,inlet}$ is changed abruptly. The principal behavior of the fluid and solid temperatures in a thermal shock test has been discussed in [11] and is shown in Figure 2. As graphically presented, the logarithmic temperature growth of solid temperatures in pre-warming process is on both sides limited by steady state operating points, with constant values of heat fluxes between fluid and solid domains.

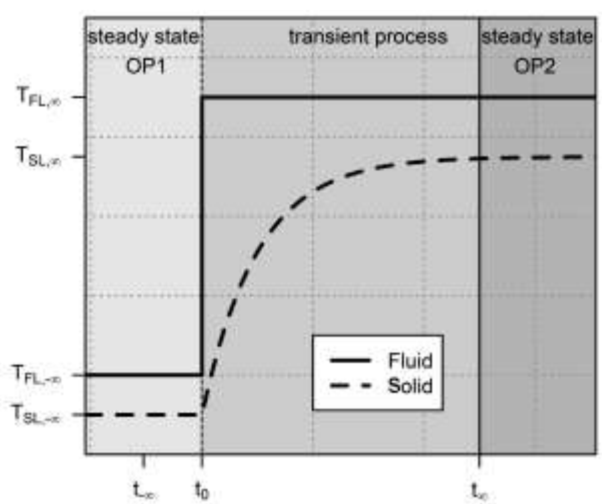


Figure 2. Qualitative idealized behavior of fluid and solid temperatures in a thermal shock test [11]

2. STEADY STATE SIMULATIONS

For numerical modelling of pre-warming process the boundary conditions at start t_0 and at the end t_∞ of procedure have to be determined (Figure 2). In this work, the cold start of pre-warming is considered, i.e. at t_0 the homogenous temperature T_0 in all solid domains is given. Therefore, simple CFD simulation with constant wall temperature and turbine inlet temperature $T_{T,inlet}$ = $T_{FL,\infty}$ is sufficient for subsequent initialization of transient calculation. For the purpose of predefinition of temperature distribution at t_0 the CHT simulations can be conducted.

In order to generate the missing boundary conditions at t_∞ (Figure 2) a mentioned steady state CHT calculation have to be carried out. The setup of this simulation corresponds to the settings of numerical model used for warm-keeping investigations and its description can be found in [10]. In warm-keeping operation (steady state *OP2*, Figure 2) a constant temperature T_R = $T(q_{Rotor,OP2})$ inside the rotor (concentrical hole) and T_H = $T(q_{Housing,OP2})$ at the outer surface of the inner steam turbine casing is defined. The temperature level is approximately 45% of the live steam temperature at nominal load.

For all steady state simulations at the inlet of the fluid state, a mass flow rate $\dot{m}$ and a constant air total temperature $T_{FL,\infty}$ (65% of the live steam temperature at nominal load) are defined as boundary conditions. A constant static pressure is used at the outlet. Due to the applied thermal boundary conditions the impact of radiation is not considered.

3. TRANSIENT SIMULATIONS

The objective of the work presented within this paper is the development of a fast and generally valid method for the calculation of the transient temperature field in turbine pre-warming with regard to thermomechanical fatigue. As shown in [12] and [13], the investigated calculation approaches may be used not only in with hot air pre-warmed turbine, but also in numerical depiction of thermal shock tests conducted in other types of turbomachinery like turbochargers. The validated numerical model of turbocharger discussed in [11], [12] and [13] has been used in present investigations in order to verify modifications introduced to extensive reference method in pursuit of calculation time reduction.

Generally, even a short literature review reveals the two main simulation approaches of transient fluid/sold heat transfer, which are commonly applied to numerical models. In the first coupled or conjugate heat transfer method (CHT), a CFD calculation is iteratively coupled with a conductive FEA calculation, which results in the solution of the energy equation at the coupled boundaries and provides the continuity of the temperatures as well as the heat fluxes at the fluid/solid interfaces. The practical application of this approach may be found in [14] and [15].

In contradiction to conjugate heat transfer, the uncoupled methods are based on the assumption of a linear relation between the convective heat fluxes across the boundary layer and the driving temperature differences. This assumption may be also understood as the requirement of constant heat transfer coefficients on the boundary interfaces. As a result, the heat transfer coefficients may be determined by means of transient CFD simulation and used as boundary conditions for a subsequent steady state FEA calculation. Examples of this approach are the heat transfer calculations of rotor stator cavities published in [16] and [17]. The main advantage of the uncoupled approach may be the significantly reduced calculation time, since no iterative solution is necessary at the fluid/solid boundary. Depending on geometry and flow boundary conditions, the uncoupled simulation approaches may lead, however, to high inaccuracies locally, mainly caused by secondary flow phenomena as well as the flow instabilities resulting in nonlinear courses of heat transfer coefficients. In this case, the accuracy of uncoupled approaches may be enhanced by means of CFD and FEA iteration loops, which iteratively increase the quality of results. This solution has been proven, however, to be inextricably linked to two main drawbacks. Firstly, the user input and probability of false setup settings obviously increase depending on the automatization level of numerical models and user experience. Secondly, the computational effort is rapidly changing with each required iteration loop.

Owing to the high calculation times, the conjugate heat transfer method is mainly used for steady state calculations. A time accurate calculation based on the coupled method requires an extreme computational effort, since the ratio of convective to conductive heat transfer timescales amount to about 10^{-4}, as reported in [18]. Consequently, simplifying assumptions have to be made to reach acceptable calculation times for a coupled method simulation of a transient heating or cooling process.

In the present paper three numerical methods are applied to a numerical model with hot air pre-warmed steam turbine in order to calculate the transient fluid/solid heat transfer. In a first step extensive simulations are conducted, in which the fluid and the solid calculations are coupled over the whole transient process. Therefore, this simulation approach belongs to a coupled methods family and will be referred to as “Frozen Flow” method (FF), as the fluid equations are partly frozen and not solved over the certain periods of time. A detailed description of this method is given in [11] and in the following sections. Additionally, a few modifications are introduced to the original FF method in order to further reduce calculation time with simultaneously almost unaffected solution accuracy. For the purpose of validation, the modified FF method is applied to mentioned numerical model of turbocharger and compared with original data.

Based on the results of the FF approach obtained from the numerical model of pre-warmed steam turbine the investigation, points P1 in vane and P2 in blade domains have been identified (Figure 1), which serve as reference for subsequent comparison. The selection criterion for the investigation points is the location of the highest temperature gradients occurring in each analyzed domain in pre-warming operation.

In second step an uncoupled approach known as “transient finite element analysis method-exponential” (TFEA-EXPO) method is used in order to model the exponential behavior of the heat transfer coefficients in a transient process. This method is based on commonly applied uncoupled approach and has been developed for purposes of thermomechanical analysis of turbocharger detailed presented in [11].

Furthermore, a third approach named Equalized Timescales method (ET) is used for conjugate heat transfer simulation of fluid and solid interaction in pre-warming operation of steam turbine. This calculation time saving approach is based on the modification of solid state properties in pursuit of the reduction of cooling or heating processes. In the following sections, all the three methods are described in detail.

3.1 FF method

The commonly used method for an analysis of the fluid/solid conjugate heat transfer is an approach based on [19] and [20]. Due to different time scales of solid and fluid state, and with regard to coupled solution for all domains in each time step, it is assumed that the pressure and the velocity distribution of the fluid remain constant over certain periods of time. With this assumption mass, momentum and turbulence equations of the fluid state do not need to be solved and much larger time steps can be applied. It has been show in [13] that in the case of thermal shock tests, which are characterized by high heat transfer rates between fluid and solid especially at the very beginning of the transient process, solving only energy equation may lead to higher inaccuracies in solution - 6% deviation in temperature gradients courses as reported in [13].

To consider the changes in the fluid field, the method described in [19], [20] may be extended by updating the fluid field in defined time points or time periods. Two main solution approaches can be distinguished. On one hand, it is possible to conduct a very short fully transient simulation in which all the equations (mass, momentum, energy and turbulence) are solved. This approach was applied to solution procedure described in [13]. On the other hand, it is also conceivable to replace the fully transient updates simulations with steady state CFD simulations. In the presented paper, the second possibility has been investigated using at first step exactly the same validated model of turbocharger, which served as basis for investigations presented in [11], [12] and [13].

The model of turbocharger used for the FF calculations purposes and for comparison between particular updated approaches comprises the turbine housing, the turbine wheel and the inlet and outlet pipes.

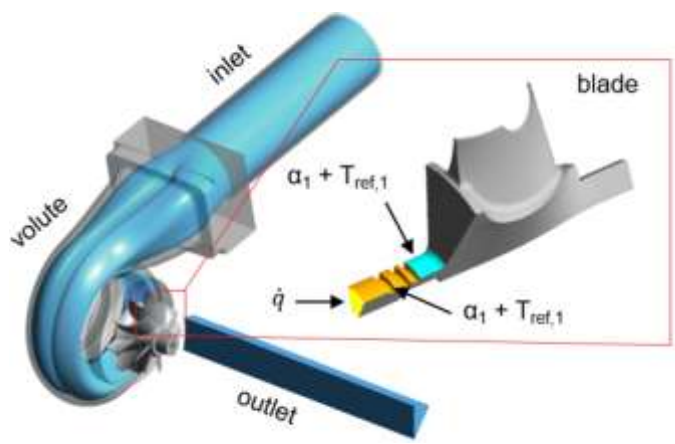


Figure 3. Numerical model of Turbine Housing (TH) and Turbine Wheel (TW)

As presented in Figure 3, the turbine wheel is modeled as a single rotor passage by means of peripheral averaging (mixing plane) at the rotor stator interfaces. The bearing housing and the compressor are not included in the model. On the boundary plane at the shaft, boundary conditions ($\alpha_1 + T_{ref,1}$, $\alpha_2 + T_{ref,2}$, q) are defined to describe the heat fluxes to the lubricant oil and to the compressor side. These boundary conditions are estimated by using experimental data. The heat shield at the turbine wheel back is assumed to be adiabatic as it is expected that the convective and radiative heat fluxes to the bearing housing are small. The mesh of the whole turbine model consists of approximately 6.2 million nodes. 3.7 million nodes are located in the fluid state and 2.5 million nodes discretize the solid body. In the fluid boundary layer, the dimensionless wall distance y+ is lower than one and the low Reynolds kω-SST turbulence model is applied. A mesh study has been conducted to ensure that the results are unaffected by the mesh. In following investigations the heating up thermal shock process with mass flow m = 0.4 kg/s and fluid temperature $T_{Fl,-\infty}$ = 200°C being abruptly changed to $T_{Fl,\infty}$ = 600°C (Figure 1) is considered. Further information about numerical setup is provided in [13].

In pursuit of model validation, the temperatures on the turbine wheel during the thermal shock test are measured. The individual positions of the type K thermocouples are shown in Figure 4. The thermocouples in the turbine wheel are connected to a transmitter, which is located at the shaft end near the compressor inlet. The transmission of amplified signals from the rotating to stationary system is done using an antenna. This antenna also supplies the transmitter with HF-energy. Additional type K thermocouples are attached on the outer surface of the turbine housing to record the temperature distribution. A more detailed description of the test rig is given in [21].

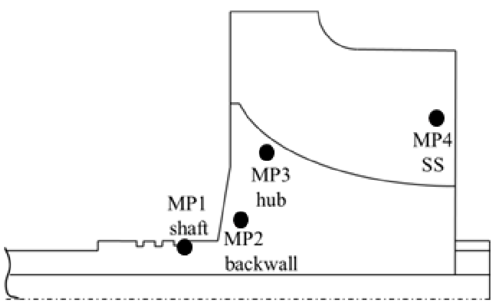


Figure 4. Position of the thermocouples at the turbine wheel

As shown in [13], the original FF method is based on fully transient flow field updates, in which all equations (mass, momentum and energy) are solved for very short periods. These short periods are hereafter referred to as “Transient Update” periods. The simplified periods are named “Only Energy” periods. In the case of turbocharger three “All Equation” periods at 0 s, 6 s, and 60 s are calculated for numerical depiction of the thermal shock process. An alternative possibility for fluid flow updates are steady state CFD simulations, which allow adjustment of pressure and velocity fields to calculated temperature values in domains. This fluid field update method is referred to as “Stationary Update”. The schematic representation of FF Stationary Update method is provided in Figure 5. At the beginning of simulation procedure, a steady state CHT All Equation 1 or CFD All Equation 1 calculation (depending on solid body initialization) is conducted. The outgoing from the generated state of fluid the transient CHT Only Energy 1 simulation (Figure 5) has to be calculated until discrete time point of update simulation has been achieved. The temperature T_{Solid} from the CHT Only Energy 1 simulation are used in the next step as the boundary conditions for subsequent CFD All Equation 2 calculation. Finally, in the CHT Only Energy 2 simulation the fluid state is being initialized by the results of CFD All Equation 2 and the temperature distribution of solid domains is set to be equal to these from CHT Only Energy 1 calculation. The presented CFD update procedure may be repeated depending on required number of fluid flow updates.

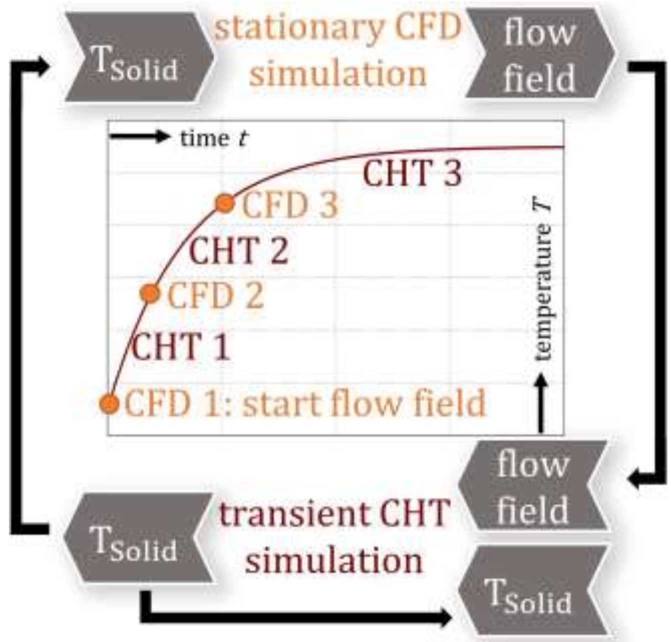


Figure 5. Schematic representation of FF Stationary Update method

In order to evaluate the impact of the updates on the fluid field the averaged relative Reynolds number for turbine wheel is determined as follows:

$$\mathrm{Re}_{\Omega,av} = \frac{w_{av}\ \rho_{av}\ D_{TW}}{\mu_{av}} \quad (1)$$

Due to turbulent flows, the heat transfer between fluid and solid is largely influenced by the convective heat transfer in the boundary layer and therefore, the averaged Reynolds-Number seems to be an apposite criterion for accuracy evaluation of the fluid simulation. The time courses of the averaged Reynolds-Number for both update calculation procedure are presented in Figure 6. The impact of the All Equation simulations on the fluid field is easily noticeable especially in the first 100 s of thermal shock, when the highest heat

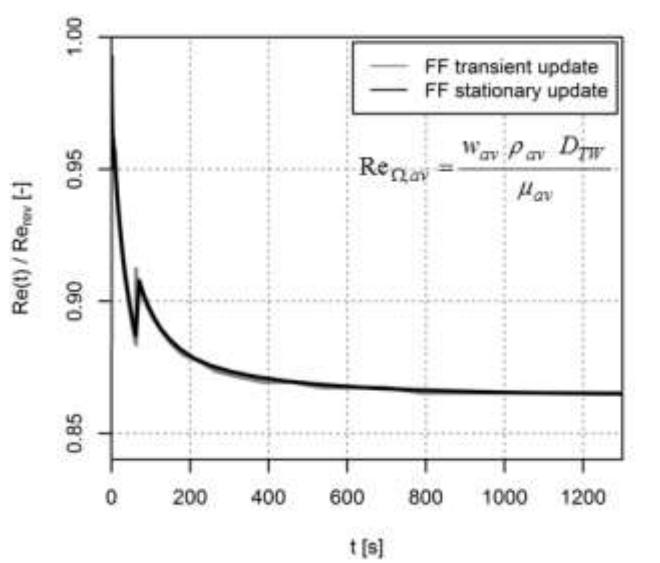


Figure 6. Averaged relative Reynolds-Number over time for a heating up process (TFl,-∞ = 200°C → 600°C at m = 0.4 kg/s)

fluxes over the boundary interfaces occur. This discontinuities in the Reynolds-Number courses are caused by the adjustment of pressure and velocity fields to the energy content. The value of discontinuities show the error of the assumption of a constant fluid field (Only Energy). As proven in [13], the committed error at the end of Only Energy calculation with four updates during first 200 s is under 1 %. In summary, expanding the Only Energy approach with additional update simulations results in higher quality of results and smaller error in averaged Reynolds-Number values – in the investigated case of turbocharger by up to 6 %. Furthermore, the FF Stationary Update calculations provide almost identical time courses of averaged Reynolds-Number as FF Transient Updates. In pursuit of FF Stationary Update method validation, the temperature values obtained from both update approaches are compared with experimental data in measuring point MP3. As depicted in Figure 7, the calculated temperature values show good agreement with experimental values. In order to evaluate the temperature deviation in all measurement points on turbine wheel, the numerical and experimental data are normalized to a time step of 1.0 s by a cubical interpolation. Thereby, a root mean square deviation is determined over a period of 800 s:

$$\Delta T_{av} = \sqrt{\frac{1}{n}\sum_{i=1}^{n}\left(T_{exp} - T_{calc}\right)^2} \qquad (2)$$

The obtained values of RMS deviations are presented in Figure 8.

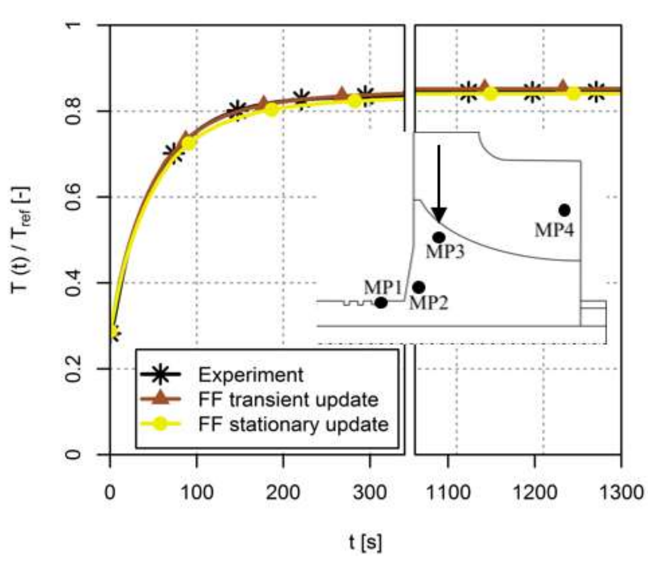


Figure 7. Temperature at measuring point 3 over time for a heating up process (TFl,-∞ = 200°C → 600°C at m = 0.4 kg/s)

The FF Transient Update method, as well as the FF Stationary Update method, provide the temperature time plots that correlate well with experimental data. Hence, regarding computational time savings, in steam turbine pre-warming numerical calculations as the reference method the FF Stationary Update method will be used.

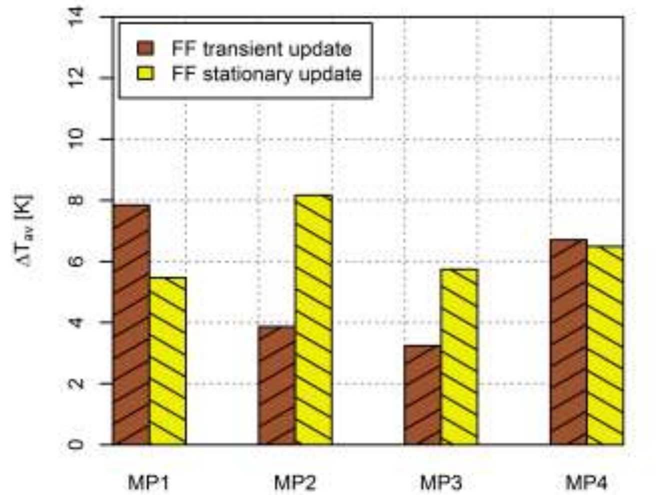


Figure 8. RMS deviation between measured and calculated temperatures at the measuring points

3.2 TFEA-EXPO

As mentioned in the previous sections, the TFEA-EXPO method is an uncoupled approach based on transient FEA simulation. In order to model the transient fluid field, heat transfer coefficients at the start and end of the transient process are derived from steady state CFD or CHT calculations. The obtained heat transfer coefficients are interpolated over the time span of transient process for each cell element. After the switch of the turbine fluid inlet temperature from $T_{Fl,-\infty}$ to $T_{Fl,\infty}$ the inlet conditions and the outlet pressure remain nearly constant during the transient process. Thereby, the changes of the fluid state result mainly from the changes of heat fluxes between fluid/solid interfaces as well as from the wall temperatures. Thus, for the interpolation of the heat transfer coefficients it is assumed, that those linearly depend on the wall temperatures. With regard to the exponential behavior of the wall temperatures this interpolation method results in an exponential time plot of the heat transfer coefficients over the time. Hence, the method is abbreviated with TFEA-EXPO. The following equations are applied to interpolate the heat transfer coefficients for each cell of the pre-warmed steam turbine:

$$\alpha_i(t) = \alpha_{i,0} + I(t)\left(\alpha_{i,\infty} - \alpha_{i,0}\right) \qquad (3)$$

$$I(t) = \left(T_i(t) - T_{i,0}\right) / \left(T_{i,\infty} - T_{i,0}\right) \qquad (4)$$

Equation 4 T_i (t) describes the time-dependent wall temperature. This temperature is directly used from the transient FEA calculation. Hence, the boundary conditions depend on the solution of the FEA simulation and are calculated iteratively in every time step. The variables $T_{i,0}$ and $T_{i,\infty}$ may be determined from steady state CHT calculations conducted respectively at the beginning and at the end of thermal shock process ($t \rightarrow \infty$). Similarly, the variable $\alpha_{i,\infty}$ in Equation 3 describes the heat transfer coefficient derived from steady state $CHT_{t\rightarrow\infty}$. Finally, the variable $\alpha_{i,0}$ refers to heat transfer coefficient at the beginning of the thermal shock process immediately after switching the inlet temperature of the turbine from $T_{Fl,-\infty}$ to $T_{Fl,\infty}$ (Figure 1). The further information about TFEA-EXPO method may be found in [12].

3.3 ET-method

The Equalized Timescales method may be classified just like FF approach as a conjugate heat transfer method. The ET method is a time saving approach, which similarly to the other presented simulation procedures may be used to calculate the transient temperature fields with regard to thermo-mechanical fatigue. Correspondingly, in this method the fluid and the solid state are calculated coupled with the whole transient process. However, the fluid calculation is not simplified like in the FF approach. Thus, a method characterized by high accuracy and small user effort is expected. The main obstacle to overcome remains unchanged: in case of a coupled calculation the ratio of timescales in fluid to timescales in solid state amounts to about 10^{-4}, as given in [18]. These different timescales cause long conductional heat transfer processes in the solid state that have to be calculated with small time steps to achieve numerical stability of the fluid calculation. In order to reduce the calculation times in this method, the timescales of fluid and solid are equalized by modifying the material properties of the solid state. Thus, this method is named Equalized Timescales method (ET).

To describe the approach of the ET method in more detail, a simple analytical approach is considered in the following sections. For a solid body with a homogenous temperature distribution, the timescale of a convectional heating or cooling process can be determined from a simple analytical equation:

$$\rho_{SL}\, c_{p,SL}\, V_{SL}\, \frac{\partial T_{SL}}{\partial t} = -A_{SL}\, \alpha \left(T_{SL} - T_{FL}\right) \tag{5}$$

With the assumptions of constant material properties, a constant heat transfer coefficient and a constant fluid temperature over the time, the solution of Equation 5 leads to Equation 6:

$$\begin{aligned} T_{SL} &= \left(T_{SL,0} - T_{FL}\right) e^{-\frac{\alpha A_{SL}}{c_{p,SL}\, \rho_{SL}\, V_{SL}} t} + T_{FL} \\ &= \left(T_{SL,0} - T_{FL}\right) e^{-\frac{t}{\tau}} + T_{FL} \end{aligned} \tag{6}$$

The exponent of the Euler's Number represents the timescale τ of the heating or the cooling process. It depends on the heat transfer coefficient α, the density ρ_{Sl} and the specific heat capacity $c_{p,Sl}$. In the ET method, the timescales of the fluid and solid are equalized by reducing the specific heat capacity of the solid state by the speed up factor SF. The modified specific heat capacity is named c_p* and is defined by the following relation:

$$c_{p,SL}^{\;*} = \frac{c_{p,SL}}{SF} \tag{7}$$

From 6 it can be derived that the reduction of the specific heat capacity results in a reduced heating or cooling time of the solid body. The modified time is named t* in the present work and is connected with the real time by the speed up factor SF for this simplified case. The definition of the modified time is given in Equation 8:

$$t^* = \frac{t}{SF} \tag{8}$$

Implementing this approach to a coupled fluid/solid heat transfer simulation results in significantly reduced calculation times as shorter conductional heating or cooling processes are calculated and the time step of the fluid calculation can remain constant. To determine the real heating or cooling time of a transient process from these simulations, the resulting time t* is multiplied by the speed up factor SF according to Equation 8.

The Fourier- and the Biot-Number are assessed in order to prove the suitability of the described approach for the determination of the thermal behavior of the solid state in a transient process. As described in [22], transient temperature fields of the same geometry and the same boundary conditions are similar if the Biot-Number and the Fourier-Number are equal. Thus, the Biot- and the Fourier-Number must remain constant when the specific heat capacity is modified. In Equation 9, the modified specific heat capacity c_p* and the modified time t* are introduced in the definition of the Fourier-Number:

$$\begin{aligned} Fo &= \frac{\lambda_{SL}\, t}{c_{p,SL}\, \rho_{SL}\, L^2} = \frac{\lambda_{SL}\, t^*\, SF}{c_{p,SL}^{\;*}\, SF\, \rho_{SL}\, L^2} \\ &= \frac{\lambda_{SL}\, t^*}{c_{p,SL}^{\;*}\, \rho_{SL}\, L^2} = Fo^* \end{aligned} \tag{9}$$

As shown, the unmodified and the modified Fourier-Numbers are identical for the described simplified approach. Unlike the Fourier-Number, the Biot-Number is independent of the specific heat capacity and of the time. However, the Biot-Number depends on the heat transfer coefficient:

$$Bi = \frac{\alpha\, L}{\lambda_{SL}} = Bi^* \tag{10}$$

With the assumption of a constant heat transfer coefficient, the Biot-Number remains constant. To investigate the assumption of a constant heat transfer coefficient in more detail and to verify the ET method in case of numerical modeling of pre-warming processes, all the methods are applied to the numerical model of pre-warmed steam turbine. For the ET method, a speed up factor of 1000 is used.

4. COMPARISON OF NUMERICAL METHODS FOR PRE-WARMING CALCULATION

The main goal that the TFEA-EXPO and the ET approaches strive for is the possibility to accurately calculate the temperature gradients resulting from 3D flow structures with significantly reduced computational and user effort in comparison with the FF method.

In pursuit of verification and comparison of the methods, a steam turbine pre-warming process modeled as thermal shock has been calculated with all three simulation approaches. It is assumed, that at the beginning of the pre-warming, all solid domains in investigated 1-stage turbine model have homogenous temperature distribution equal to $T_{SL,-\infty}$. Similarly to thermal shock process in the discussed turbocharger, the fluid inlet temperature is being abruptly changed from $T_{FL,-\infty}$ to $T_{FL,\infty}$ at time point t_0.

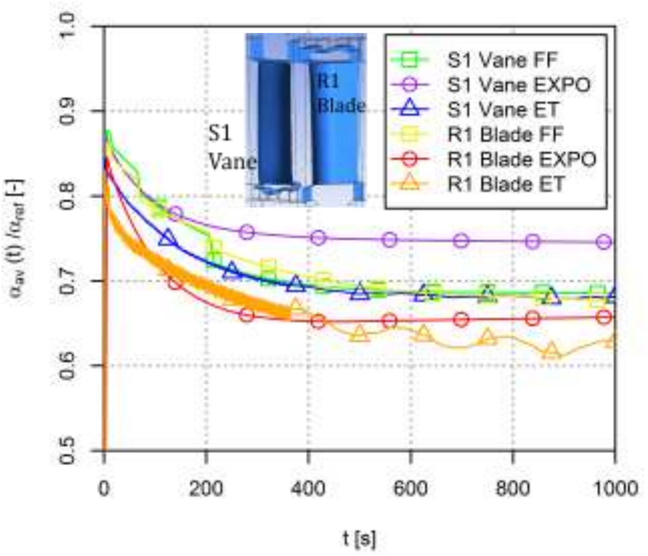


Figure 9. Heat transfer coefficients on the surface of vane and blade

In first step the heat transfer coefficients are derived from the FF calculations and used for subsequent comparison with HTCs obtained from the TFEA-EXPO as well as from the ET simulation. The area averaged heat transfer coefficients on the surface of the vane and blade are depicted in Figure 9. As expected, all the plots of the heat transfer coefficients prove an exponential behavior.
The oscillating plot of HTCs on blade indicates, that the Speed up factor in the ET method cannot be increased without any further limitations. These oscillations result from unsteady flow effects which can have a large influence on the solid temperatures for higher speed up factors due to the similar timescales of fluid and solid state.

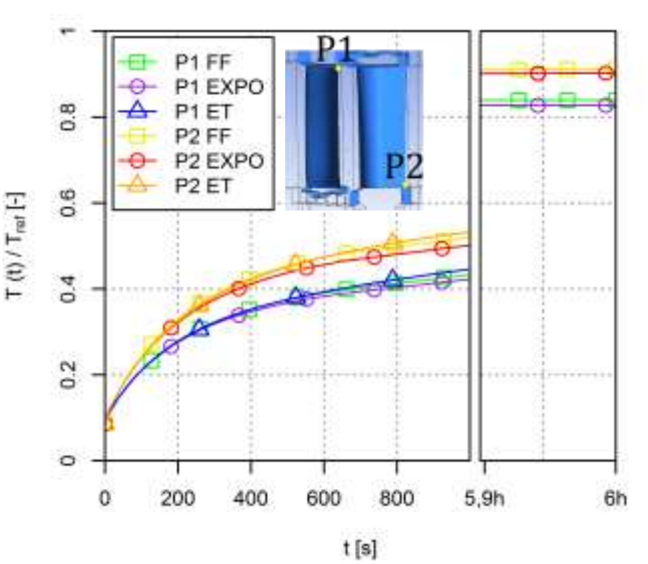
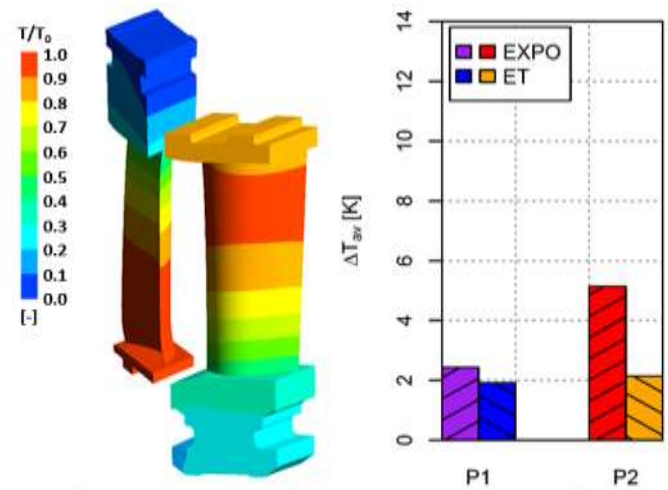


Figure 10. Temperatures calculated at characteristic points P1 and P2

In a second step, the calculated temperatures at the investigation points P1 and P2 are compared with reference approach. As shown in Figure 10 all numerical methods provide temperature plots which show a good agreement with the reference FF method. In order to quantify the respective error, the root mean square (RMS) deviation between measured and calculated temperatures is determined over a period of the first 1000 s for a time step of 1.0 s. The RMS deviations for both investigation points P1 and P2, comparison of the computing times (calculated for first 1000 s of pre-warming procedure) as well as the temperature distribution in investigated single stage turbine model are presented in Figure 11.



method	boundary conditions and initiations files		transient calculation		summary
	CFD start	CHT end	transient parts	CFD updates	
FF	66	24	360	131	581
ET	66	24	2658		2748
EXPO	66	24	6		96

Figure 11. Temperature distribution, RMS deviation and overview over the calculation times in [core-h]

5. CONCLUSION

In the present work two calculation methods – TFEA-EXPO and ET for modeling of transient temperature fields in pre-warmed steam turbine are conducted and compared with an extensive calculation approach for verification purposes. Additionally, some changes resulting in reduced computational time are introduced into numerical procedure of an extensive FF method. Moreover, in a subsequent step the modified FF method is validated against the original FF approach and experimental data using known from several other publications presenting the numerical model of turbocharger.
To conclude, all the methods presented in the present work are suitable to calculate the transient temperature field with regard to thermomechanical fatigue within the timescales of a standard design process. Furthermore, the additional savings in computational time have been achieved – 89% in case of TFEA-EXPO approach. The ET method has been analyzed only for one value of speed up factor (SF = 1000) and the good accuracy with simultaneously minimum user effort has been achieved. In future research the more computing time-efficient ET simulations with higher values of speed up factor should be investigated.

ACKNOWLEDGEMENTS

The present work is a part of the joint research program COOREFELX-Turbo in the frame of AG Turbo. This work is supported by German ministry of economy and technology under grant number 03ET7041B. The authors would like to gratefully acknowledge GE Electric whose support was essential for the presented investigations. The responsibility for the content of this publication lies with the corresponding authors.

NOMENCLATURE

A	Area
Bi	Biot-Number
Cp	Specific Heat Capacity
D_{TW}	Diameter of the turbine wheel
Fo	Fourier-Number
I	Interpolation function
L	Characteristic length
$\dot{m}$	Mass flow rate
$\dot{q}$	Heat flux rate
OP	Operating point
SF	Speed up factor
T	Temperature
t	Time
V	Volume
w	Velocity in relative frame
y^+	Dimensionless wall distance
α	Heat transfer coefficient
ΔT	Temperature difference
λ	Thermal conductivity
μ	Dynamic viscosity
ρ	Density
τ	Time scale
$\lvert\nabla T\rvert$	Resulting temperature gradient

Indices

0	Time point zero
$-\infty$	Time point before the start of a thermal shock process
∞	Time point at the end of a thermal shock
av	Averaged
FL	Fluid
H	Housing
MP	Measuring point turbine wheel
R	Rotor
ref	Reference
SL	Solid

Abbreviations

CFD	Computational fluid dynamics
CHT	Conjugate Heat Transfer
Exp	Experimental
ET	Equalized timescales
FEA	Finite element analysis
HTC	Heat transfer coefficient
RMS	Root mean square

REFERENCES

[1] J. Vogt, T. Schaaf, W. F. D. Mohr, and K. Helbig, "Flexibility improvement of the steam turbine of Conventional or CCPP," 2013.

[2] J. Klimstra, M. Hotakainen, "Smart Power Generation," 3rd improved edition, Avain Publishers Helsinki,

[3] H. Wirth, "Recent Facts about Photovoltaics in Germany," *Fraunhofer ISE*, 2016

[4] J. Spelling, M. Jöcker, and A. Martin, "Thermal Modeling of a Solar Steam Turbine With a Forcus on Start-Up Time Reduction," *ASME - Journal of Engineering for Gas Turbines and Power*, 2012.

[5] M. Topel, M. Genrup, M. Joecker, and J. Spelling, "Operational Improvements for Startup Time Reduction in Solar Steam Turbines," *ASME - Journal of Engineering for Gas Turbines and Power*, 2015.

[6] H. B. Evers, "Strömungsformen im Ventilationsbetrieb einer ein- und mehrstufig beschaufelten Modellturbine," Dissertation, Fakultät für Maschinenwesen, Universität Hannover, 1985.

[7] N. Herzog, "Numerische Untersuchung von Schwachlastströmungen in mehrstufigen Axialturbinen," Dissertation, Universität Hannover, 2008.

[8] M. Binner, J. R. Seume, "Flow Patterns in High Pressure Steam Turbines During Low-Load Operation," *ASME - Journal of Turbomachinery*, no. 136, 2014.

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

[10] D. Toebben, P. Łuczyński, M. Diefenthal, M. Wirsum, S. Reitschmidt, W. F. D. Mohr and K. Helbig, "Numerical Investigation of the Heat Transfer and Flow Phenomena in an IP Steam Turbine in Warm-Keeping Operation with Hot Air," Proceedings of *ASME Turbo Expo* GT2017-63555, Charlotte, USA, 2017

[11] M. Diefenthal, P. Łuczyński, C. Rakut, M. Wirsum and T. Heuer, "Thermomechanical Analysis of Transient Temperatures in a Radial Turbine Wheel," *ASME - Journal of Turbomachinery*, no. 139, 2017

[12] M. Diefenthal, P. Łuczyński and M. Wirsum, "Speed-up Methods for the Modeling of Transient Temperatures with regard to Thermal and Thermomechanical Fatigue," *12th European Turbomachinery Conference*, Paper ETC2017-160, Stockholm, Sweden, 2017

[13] 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

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

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

[16] 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

[17] 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

[18] L. He, M. L. G. Oldfield, "Unsteady Conjugate Heat Transfer Modeling," *ASME Journal of Turbomachinery*, 133, No. 3, July, 2011

[19] T. Heuer, B. Engels, A. Klein, and H. Heger, "Numerical and Experimental Analysis of the Thermo-Mechanical Load on Turbine Wheels of Turbochargers," *ASME Turbo Expo* GT2006-90526, Barcelona, Spain, 2006

[20] T. Heuer, B. Engels, P. Wollscheid, „Thermomechanical analysis of a turbocharger based on conjugate heat transfer," *ASME Turbo Expo* GT2005-68059, Reno-Tahoe, USA, 2005

[21] H. Tadesse, C. Rakut, M. Diefenthal, M. Wirsum, and T. Heuer, "Experimental investigation of steady state and transient heat transfer in a radial turbine wheel of a turbocharger," *ASME Turbo Expo* GT2015-43165, Montreal, Canada, 2015

[22] H. D. Baehr, and K. Stephan, „Wärme- und Stoffübertragung," 7th ed., *Springer*, Heidelberg, Germany, 2010