ABSTRACT

Recently, the reduction of steam turbine start-up times has become increasingly important. In pursuit of this objective, General Electric has developed a concept for both the pre-warming and warm-keeping of steam turbines with hot air. To investigate the thermally induced stresses, a computationally efficient method is required for the simulation of transient temperature distribution in the turbine during pre-warming. In the presented work, four transient calculation methods are investigated for pre-warming simulation. Firstly, a modified Frozen Flow method is used in order to conduct transient conjugate heat transfer (CHT) simulations. Secondly, two uncoupled approaches called TFEA-LINR and the TFEA-EXPO are applied to calculate heat transfer coefficients from steady-state calculations. In addition, a fourth approach called the Equalized Timescales (ET) method is presented. In this CHT method, the specific heat capacity of the solid state is reduced by a “speed up factor” to reduce transient heating time.

1. INTRODUCTION

The situation of the European energy market is evolving dynamically. The greatest challenge engineers have to face in the field of successful integration of renewable energy sources with existing power systems is a variability and certain unpredictability of electricity generation from wind or solar energy sources.

The flexibility and load gradients of coal-fired power plants in particular must be increased for both economic reasons, and in order to ensure a balance between total generation and consumption of power in real time. The typical start-up time of a conventional power unit (over 500 MWa) from cold state is estimated to be about five to eight hours (Vogt et al., 2013), and is influenced by high values of thermal stresses induced in the thick-walled HP and IP steam turbine components. Steeper load gradients during the frequent start-ups of a conventional power plant unavoidably lead to higher thermal stresses and, from a lifetime assessment point of view, increase the probability of crack development. Hence, a reduction in the start-up time with an unaffected turbine lifetime consumption requires innovative technical solutions.

A possible solution to achieve a reduction of thermal stresses during start-up is the concept of turbine warm-keeping, or pre-warming. The warm-keeping methods focus on minimizing or compensating for heat losses that occur during periods of turbine inactivity and, in contrast to pre-warming operations, do not require arduous turbine heating from cold states.

The concept of warm-keeping or pre-warming of steam turbines with hot air (depending on the initial thermal boundary conditions) was presented by Helbig et al. (2014). In the proposed technical arrangement, the rotating turbine is heated by circulated hot air. The first investigation of warm-keeping operations concerning fluid field as well as heat transfer phenomena (based on the concept developed by Helbig et al. (2014)) was given by Toebben et al. (2017). This research is based on steady state calculation approaches. In the next step presented in this work, the numerical methods allowing unsteady simulation of pre-warming operations are investigated.

In the presented paper, the pre-warming operation is modelled as a thermal shock process, known from the investigation of time-dependent temperature fields in turbochargers described in (Diefenthal et al., 2015) and (Diefenthal et al., 2017a). In the thermal shock procedure, the turbine fluid inlet temperature is abruptly changed in a step function at time point $t_0$ from $T_{FL,\infty}$ to $T_{FL,\infty}$, as graphically...
depicted in Figure 1. The resulting logarithmic growth of solid body temperatures in pre-warming processes is situated between cold turbine state at operating point 1 (OP1) and warm-keeping operation at operating point 2 (OP2). Thus, due to the intense heat exchange between cold turbine components and hot working fluid, especially at the first phase of the pre-warming procedure, higher temperature gradients are expected as in warm-keeping operation. The main objective of the research presented in this paper is the development of a fast and generally valid method for the calculation of the transient temperature fields in turbine pre-warming, with regards to thermomechanical fatigue. In order to achieve this goal, several unsteady simulation approaches known from literature are implemented into numerical models of pre-warmed steam turbines. Of particular interest are the numerical methods that enable the bridging of the gap between vastly different time scales of solid and fluid states.

![Figure 1](image1.png)

**Figure 1** Behavior of fluid and solid temperatures in pre-warming and warm-keeping operation

In the following section (Sec. 2), the numerical models used for steady and transient calculations are presented. Subsequently, the unsteady simulation approaches allowing the calculation of transient temperature fields in fluid and solid states are discussed (Sec. 3). The comparison of the results obtained using the chosen simulation methods is provided in the penultimate section (Sec. 4), just before the conclusion (Sec. 5).

2. NUMERICAL MODEL

In cooperation with the industrial partner General Electric, a numerical model of a single repeating turbine stage has been developed. For pre-warming calculation purposes, the numerical model for warm-keeping operation, known from research presented by Toebben et al. (2017), is used, and hence, only the most relevant information concerning model setup and changes introduced to original settings are provided. Due to the high live steam turbine temperature, the thermally induced stresses arise particularly within the first turbine stage. Thus, the investigation of heat transfer and calculation approaches focuses on a single stage model. As presented in Figure 2, one blade passage (approx. 2.5°) of the turbine wheel is modelled by means of circumferential periodicity. The hexahedral mesh of all single stage turbines consists of approximately 6.8 million nodes. 3.6 million nodes are located in the fluid state and 3.1 million nodes discretize the solid body. In the fluid boundary layer the dimensionless wall distance $y^+$ is lower than 1. A mesh study has been conducted by Toebben et al. (2017) to ensure that the results are unaffected by the mesh quality. To model the turbulence in thermal sub-layers, the low Reynolds kω-SST turbulence model is used. Furthermore, the fluid state is assumed to be a fully turbulent ideal gas with constant fluid properties. Constant material properties are also applied for the solid. The thermal boundary conditions on the outer surfaces of the rotor and housing differ between steady and transient simulations, and thus are presented in the following sections. The axial walls of the model are considered to be adiabatic ($q=0$ W m⁻², cp. Figure 2).

![Figure 2](image2.png)

**Figure 2** Numerical model with marked boundary conditions and used denotation

2.1 Steady state simulations

For generation of the boundary conditions at the start $t_0$ and end $t_e$ of the pre-warming process (cp. Figure 1), the steady state CHT simulations must be calculated. In the following investigations, the so called cold start is being considered. Therefore, the temperature distribution at $t_0$ in solid domains is homogenous and does not have to be determined numerically. Hence, only simple CFD with constant predefined wall temperatures and turbine inlet temperature $T_{TI}=T_{FL,∞}$ is required to initialise the fluid field of transient simulations. Furthermore, for definition of end boundary conditions for fluid and solid state at time $t_e$, the steady state CHT calculations have to be conducted (Toebben et al., 2017).

In the case of steady state calculations, the transformation from the non-rotating to the rotating system is performed by Frozen Rotor Interface, in order to capture the strong backflows through the domains interfaces. The fluid field analysis presented by Toebben et al. (2017) reveals that in warm-keeping operations the turbine flow is characterized by windage effects. This implies that for both the discussed numerical setup and, interestingly, for multistage warm-keeping investigations, unsteady approaches may be required. Nevertheless, a satisfying convergence level (defined by presupposed values of residuals and imbalances) in
single stage simulations is achieved by a low value of timescale applied to fluid domains, which result in under-relaxation of governing equations.

Moreover, the missing thermal boundary conditions for rotor and housing are defined as follows: surfaces of the predefined concentric hole in the rotor, and the outer surfaces of the inner steam turbine casing, are represented by $T_R$ and $T_H$ respectively (cp. Figure 2). The temperature values amount to approximately 45% of the live steam temperature at nominal load.

2.2 Transient simulations

The pre-warming operation presented in this work is modelled as a thermal shock process. Therefore, corresponding with previously described assumptions of the thermal shock procedure, the turbine inlet temperature $T_{TI}$ in pre-warming operation is considered to be constant and equal to $T_{ILC}$. As a result of this assumption, the numerically determined temperature gradients in turbine components can be overestimated, especially at the beginning of the thermal shock process. Nevertheless, the chosen modelling approach allows the identification of critical zones in a pre-warmed steam turbine where, due to the varying heat exchange conditions in the flow channel, high thermally induced stresses occur.

The temperatures on the outer surfaces of the rotor $T_R = T_{(Hugh)}$ and housing $T_H = T_{(Hhousing)}$ (cp. Figure 2) are defined based on the values of heat flux obtained from steady state calculations. At the end of pre-warming procedure - time point $t_{0}$ - the boundary conditions correspond to the boundary conditions of warm-keeping operation. Therefore, some of the aforementioned considerations analysing unsteady numerical approaches can additionally be used to investigate the warm-keeping operation (cp. Figure 2, OP2).

3 MODELLING OF HEAT TRANSFER

The key issue in the assessment of turbine lifetime consumption, as well as in the determination of thermomechanical fatigue caused by high thermally induced stresses, is the modelling of heat exchange between fluid flow and solid bodies with sufficient accuracy. Two main calculation approaches can be distinguished in literature. In the coupled method, also known as the conjugate heat transfer approach (CHT), a CFD simulation is iteratively coupled with a conductive finite element analysis (FEA). The solution of the energy equation at the coupled boundaries ensures the continuity of the heat fluxes and temperature fields on the fluid/solid interfaces. The description of the investigation of employed CHT approach is given at (Bohn et al., 2003), and its technical implementation into the model of a turbocharger can be found at (Bohn et al., 2005).

In contrast to CHT methods, most of the uncoupled approaches are based on the assumption of constant heat transfer coefficients (HTC) on boundary interfaces, implying a linear relation between the convective heat fluxes across the fluid/solid interface and the driving temperature differences. The HTC values obtained by the CFD calculation are used in the subsequent step for FEA calculation. Although the uncoupled methods may lead to inaccuracies caused by locally nonlinear plots of HTC, a significant reduction of computational time can be achieved. Examples of these methods in the context of rotor-stator cavities investigation are given by Alizadeh et al. (2007) as well as Lewis and Provins (2004).

The main challenge unsteady coupled calculation approaches face are the vastly different time scales of the convective and the conductive heat transfer in fluid and solid states, as reported by He and Oldfield (2011). Therefore, the CHT methods are usually applied to steady state simulations, and their implementation into time accurate calculations requires additional assumptions.

In this work, four numerical methods – Frozen Flow (FF), TFEA-EXPO, TFEA-LINR and Equalized Timescales (ET) – are employed in order to calculate the heat transfer in pre-warming operations. Furthermore, some of the presented calculation approaches are successfully utilized by Diefenthal et al. (2017a) for modelling of heat transfer in commercial vehicle turbochargers. The fluid field in the turbine wheel of the considered turbocharger is characterized by values of Re-number between $7*10^4-2.3*10^5$ (depending on the operating point), and Pr-number of about 0.57-0.7. The heat transfer on a turbine wheel is described by Nu-numbers, the values of which vary between 61 and 204 in stationary operation. Interestingly, the numerical simulations of the pre-warming turbine operation reveal Re-numbers of about $5.0*10^5$ (value is strongly dependent on the mass flow rate), Pr-number equal to 0.72 and Nu-numbers on blade surfaces of about 67-100. Furthermore, as the concept of pre-warming of steam turbines with hot air is an innovative approach, there are still no validation data provided. Hence, due to the similar heat transfer conditions in the numerical models of both turbomachines, the modified reference simulation method of turbocharger investigations (known as Frozen Flow method) is also considered as a reference approach for pre-warming calculations. The difference between the original FF method (named as “FF transient update”) and modified FF approach (“FF stationary update”) newly presented in this work, is the way of “updating” the fluid field. In order to ensure that the introduced changes to the FF method (which result in additional computational time savings) do not significantly affect the accuracy of the original simulation approach, in the first step the FF stationary update method is validated on the model of the turbocharger. In the second step, the validated FF stationary update is used as a reference approach for pre-warming calculations. A detailed description of both the FF methods and validation of the FF stationary update approach on the model of the turbocharger is given in Sec. 3.1. Moreover, based on the FF results, the position of investigation points P1 and P2, in which the highest temperature gradients in pre-warming operation occur, is determined (cp. Figure 2). In the following sections, both points located on vane and blade surfaces are used to compare analysed simulation methods.

In the next step, the two methods belonging to the family of uncoupled heat exchange simulation approaches are investigated. In the first uncoupled approach model, the heat transfer coefficients demonstrate exponential behaviour in the thermal shock procedure and, hence, it is called a transient finite element analysis method-exponential (TFEA-EXPO). The TFEA-EXPO calculation approach has been developed by Diefenthal et al. (2015) based on another uncoupled method, which enables
determination of transient temperature fields in solid state by means of linear interpolation of HTCs over a specified time period. Therefore, the second uncoupled approach is abbreviated as TFEA-LINR. Both TFEA methods are discussed in Sec. 3.2.

The fourth and final method achieves a significant computational time reduction by modification of material properties of solid state, and thus equalization of fluid and solid time scales. Hence, this approach – which is already used for unsteady investigations of turbochargers (Diefenthal et al., 2017a; Diefenthal et al., 2017b) – is referred to as the Equalized Timescales method. This approach does not employ any assumptions concerning fluid state (like FF method), and allows coupled simulation over the whole transient process. A more detailed description of ET approach follows in Sec. 3.3.

### 3.1 FF method

In order to overcome the extremely high computational times in unsteady CHT calculations, it is assumed in the first method that the pressure and velocity distribution of the fluid field remain constant over certain periods of time. Hence, corresponding to this assumption, the mass, momentum and turbulence equations of the fluid state are not solved and consequently, larger time steps can be used. Obviously, this assumption leads to inaccuracies in solutions, especially in the first phase of the thermal shock process, mainly due to the intense heat transfer over the fluid/solid interfaces. In the FF approach originally developed for investigations of turbochargers presented by Heuer et al. (2006), no further precautions have been taken for enhancement of calculation accuracy. An improved accuracy of the discussed method has been achieved by introducing additional CHT simulations, in which all equations (mass, momentum and turbulence) are solved at the predefined time points of transient processes and the “update” of fluid field occurs (Diefenthal et al., 2015). The adjustment of velocity and pressure distribution to transient temperature fields may also be realized by replacing the fully unsteady CHT calculations with simple stationary CFD simulations. In the following, the second possibility of the fluid field update is investigated and in the first step it is applied to the turbocharger model described by Diefenthal et al. (2015) for validation purposes. Subsequently, the CFD update based FF method is employed to determine the heat exchange and flow field in a pre-warmed steam turbine.

The main goals of the experimental and numerical investigations conducted for commercial vehicle turbochargers were similar to the pre-warming objectives, as the turbocharger analysis focused on the capturing of transient temperature fields in the context of thermomechanical fatigue. The numerical model of a turbocharger with its main components including inlet/outlet pipes, volute and turbine wheel is shown in Figure 3.

The turbine wheel of the turbocharger is established as a single rotor passage based on the mixing plane assumption between stationary and rotating domains. The boundary conditions on the shaft, including heat fluxes to the gas, lubricant oil and to compressor side, are defined by means of heat transfer coefficients \( \alpha_1 \) and \( \alpha_2 \) (with reference temperatures \( T_{ref,1} \) and \( T_{ref,2} \)) and heat flux \( \dot{q} \). Similar to the steam turbine simulations, the low Reynolds k-\( \omega \)-SST turbulence model is used to calculate the flow field and heat transfer on the 6.2 million nodes mesh with dimensionless wall distance \( y^+ \) lower than 1. In the case of turbocharger investigations, the thermal shock process is also considered, with the fluid flow temperatures \( T_{FL} \) being abruptly changed from \( T_{FL,\infty} = 200 \)°C to \( T_{FL,\infty} = 600 \)°C. During the entire thermal shock procedure, the temperature values at the measurement points presented in Figure 4 are recorded. More detailed information concerning experimental as well as numerical setup may be found at (Diefenthal et al., 2015) and (Diefenthal et al., 2017a).

In the original FF method applied to the turbocharger, the fully transient CHT updates are conducted at 0s, 6s and 60s of the thermal shock process (cp. Diefenthal et al., 2015). In the presented work, the second, previously mentioned approach based on the CFD update simulation is analysed.

#### Figure 3 Numerical model of turbine housing (TH) and turbine wheel (TW)

![Numerical model of turbine housing (TH) and turbine wheel (TW)](image)

The stationary CFD update simulation procedure of the thermal shock is given in Figure 5. At the beginning of the simulation process, a conventional steady state CFD or CHT calculation is conducted for initialisation of fluid field and eventually, temperature distribution in solid state. After providing the necessary start conditions, the time-marching CHT calculation solving only energy equations (“Only Energy” period) is performed. The determined temperatures on the flow channel walls are used in steady state CFD update simulations as boundary conditions, in order to achieve the adjustment of pressure and velocity flow fields with regards to changing energy content. The fluid field obtained in CFD, together with the solid-state temperature distribution known from the last time point of the previous Only Energy CHT calculation, are sufficient for initialisation of the next Only Energy transient period. The
described procedure may be repeated for any required number of update calculations. In contrast to the original FF approach, the fully transient CHT periods are replaced with steady state CFDs. Hence, in the discussed turbocharger model, the computational time is decreased by a factor of 40 in reference to the fully transient update method. This result is of particular importance, especially in the case of simulations of long processes (like pre-}

- The changes in fluid field at the end of the thermal shock process are negligible (cp. fully transient simulation procedure “5 updates”, Figure 6).

![Figure 5 Stationary CFD update simulation procedure](image)

Figure 5 Stationary CFD update simulation procedure

 warming), which may be characterized by a higher number of required update calculations.

The heat exchange between fluid and solid in the operation of turbochargers is mainly characterized by convective heat transfer. Therefore, the impact of the updates on the fluid field in the turbine wheel is evaluated by means of averaged relative circumferential Reynolds-number, defined as:

\[ Re_{\Omega,av} = \frac{w_{av} \rho_{av} D_{TW}}{\mu_{av}} \]  

The plots of averaged Re-number against time are given in Figure 6. The impact of the transient CHT as well as the stationary CFD updates can be seen in discontinuities of the Reynolds number. The calculation errors caused by the assumption of constant fluid field – only energy approach – are indicated by the discontinuities, and have been determined for both update approaches (cp. Figure 6). The number and position of update simulations in numerical thermal shock modelling have been investigated in detail by Diefenthal et al. (2015). Concerning the solution accuracy, in addition to the computational time and user effort, three transient CHT update simulations at 0s, 6s, 60s have been chosen. Thus, the new FF modelling of thermal shock based on the steady state CFD simulations is conducted for the same time periods. As presented in Figure 6, the determined errors of discontinuities in the Re-number plot remain almost unchanged for both update procedures. Corresponding to the analysis given at (Diefenthal et al., 2015), the following conclusions can be drawn:

- The assumption of constant fluid field over the whole thermal shock procedure is not justifiable (cp. plot “0 updates”, Figure 6),
- The highest discontinuities in the plots of Re-numbers occur in the first few seconds of the thermal shock process. Hence, the adjustment of velocity and pressure values to energy field in turbine flow is of significant importance during the first 200s of the simulation procedure,

![Figure 6 Averaged relative Re-number in turbine wheel during the thermal shock process](image)

Figure 6 Averaged relative Re-number in turbine wheel during the thermal shock process

Furthermore, for evaluation purposes of time-dependent temperature fields in the turbine wheel of a turbocharger, the values of temperatures in measurement point 3 (MP3), obtained by means of both FF approaches, are compared with experimental data.

![Figure 7 Temperatures at measurement point 3 during the thermal shock process](image)

Figure 7 Temperatures at measurement point 3 during the thermal shock process

As given in Figure 7, the calculated temperatures remain in good agreement with experimental values. A more detailed assessment of temperature deviations in each of four MPs is determined using cubical interpolation of data to a time step of 1s
and the definition of relative root mean square (RRMS) deviation over a period of 800s:

$$\Delta T_{\text{RRMS}} = \sqrt{n^{-1}\sum_{i=1}^{n}(\frac{T_{\text{exp},i} - T_{\text{sim},i}}{T_{\text{exp},i}})^2}$$  \quad (2)

Figure 8 RRMS deviation between measured and calculated temperature values

The RRMS values are summarized in Figure 8. The presented analysis confirms the accuracy of FF method based on the CFD update simulations. The maximal RRMS is obtained at MP1, but its value does not exceed 4%. Hence, the developed FF approach is used as a reference method for investigations of pre-warmed steam turbines.

3.2 TFEA method

In accordance with the previous considerations, the TFEA methods are based on the uncoupled approach. The transient temperature fields are calculated by means of unsteady FEA simulation. In this paper, two main calculation approaches are considered: TFEA-LINR and TFEA-EXPO methods. In the first approach, the HTCs are linearly interpolated by the user defined time span of transient calculation, and are then held constant for the rest of simulation. Considering the exponential behaviour of HTCs in the pre-warming process, a linear interpolation depends on user experience and may lead to higher inaccuracies. The alternative approach – TFEA-EXPO – assumes that the HTCs are changing linearly to the wall temperatures. The values of HTCs are interpolated between the start and end of the pre-warming procedure:

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

$$I(t) = \frac{T_i(t) - T_{i,0}}{(T_{i,\infty} - T_{i,0})}$$  \quad (4)

The wall temperatures $T_{i,0}$ and $T_{i,\infty}$ as well as the heat transfer coefficients $\alpha_{i,0}$ and $\alpha_{i,\infty}$ can be determined from steady state CHT simulations conducted at the beginning $t = t_0$ and at the end $t = t_\infty$ of the thermal shock process (cp. Figure 1). The detailed description of TFEA-EXPO method is given at (Diefenthal et al., 2015).

3.3 ET method

According to the investigation presented by He and Oldfield (2011), the timescales of solid to fluid state differ by a factor of about 10^4. Hence, in conventional unsteady CHT simulations, the long conduction process must be calculated with small time steps resulting from numerical stability of the fluid state. As shown in (Diefenthal et al., 2017a) and (Diefenthal et al., 2017b), the problem of the different timescales in both states can be solved by modification of solid domain properties. The reduction of specific heat capacity in solid state by a speed up factor SF leads to a significantly faster heating or cooling process:

$$C_{P_{\text{SL}}}^* = \frac{C_{P_{\text{SL}}}}{SF} \quad t^* = t/SF$$  \quad (5)

The modified specific heat capacity $C_{P_{\text{SL}}}^*$, and resulting time of transient process $t^*$, depends on the value of SF, which can be arbitrarily chosen at the beginning of simulations. Although considering computational time the SF should be as high as possible, the value of $10^4$ stated previously may lead to an unphysical solution, in which solid domains react faster to changes in boundary conditions than the fluid state. Additional questions requiring more detailed analysis are the interactions of SF with time step (TS) on results accuracy during the first phase of pre-warming processes, which is characterized by intense heat exchange, and in warm-keeping operation.

4 COMPARISON OF NUMERICAL METHODS FOR MODELLING OF PRE-WARMING PROCESS

All four previously discussed methods are applied to the numerical model of a pre-warmed steam turbine in order to determine the transient temperature fields in solid domains with regard to thermomechanical fatigue. The overview of the conducted simulations, as well as the computational times of particular simulations, are provided in Table 1.

<table>
<thead>
<tr>
<th>method</th>
<th>boundary conditions and initialisation files</th>
<th>transient calculation</th>
<th>summary</th>
</tr>
</thead>
<tbody>
<tr>
<td>FF</td>
<td>CFD at start CHT at end transient parts CFD updates</td>
<td>65.7 23.7 359.8 131.5</td>
<td>580.7</td>
</tr>
<tr>
<td>EXPO</td>
<td>65.7 23.7</td>
<td>5.9</td>
<td>95.3</td>
</tr>
<tr>
<td>LINEAR</td>
<td>65.7 23.7</td>
<td>7.4</td>
<td>96.8</td>
</tr>
<tr>
<td>SF10°TS10^4</td>
<td>65.7 23.7</td>
<td>2657.9</td>
<td>2747.3</td>
</tr>
<tr>
<td>SF10°TS10^5</td>
<td>65.7 23.7</td>
<td>2434.0</td>
<td>2523.4</td>
</tr>
<tr>
<td>SF20°TS10^4</td>
<td>65.7 23.7</td>
<td>373.3</td>
<td>462.7</td>
</tr>
<tr>
<td>SF20°TS10^5</td>
<td>65.7 23.7</td>
<td>273.3</td>
<td>116.7</td>
</tr>
<tr>
<td>SF10°TS10^6</td>
<td>65.7 23.7</td>
<td>135.6</td>
<td>225.2</td>
</tr>
<tr>
<td>SF20°TS10^7</td>
<td>65.7 23.7</td>
<td>27.1</td>
<td>116.5</td>
</tr>
</tbody>
</table>

Table 1 Comparison of the calculation times of the different methods in core-h.

Due to several applications known from literature, and in correspondence with the investigations based on the validated numerical turbocharger model, the FF method is chosen as a reference approach. Furthermore, the impact of SF in the ET method on the quality of the results is tested in simulations with SF 1000, 10 000, 20 000, 100 000 and TS equal to 0.001s. Additionally, the influence of TS is additionally investigated for a constant value of SF 10 000. In order to assess the differences in transient temperature fields determined by the various presented numerical approaches, the time-dependent plots of temperatures and of temperature gradients in characteristic points P1 (vane) and P2 (blade)
Temperatures calculated at characteristic points P1 (vane, above) and P2 (blade, below) (cp. Figure 2) are provided in Figure 9. As expected, the highest deviations between investigated approaches and the reference method occur during the first 400s of the pre-warming process, mainly due to the high heat fluxes across fluid/solid boundaries. Interestingly, in warm-keeping operations all approaches lead to similar results. Moreover, the uncoupled methods deliver very good approximations of transient temperature fields (compared to the reference method) in pre-warmed turbines.

Surprisingly, the TFEA-LINR method achieves slightly better accuracy than the TFEA-EXPO approach, mainly due to the appropriately selected interpolation time. In the case of CHT simulations – the ET method – a distinct trend in temperature plots is observed: the lower value of SF and TS, the higher the quality of the solution. Hence, a very good agreement with the FF method is obtained by means of ET calculation with SF 1000 and TS 0.001s. The plots of Bi-numbers on vane and blade surfaces given in Figure 10 confirm the above statements. Particularly interesting are the highly underestimated values of Bi-numbers, and thus similarly underestimated values of HTCs, determined by simulations with SF higher than 10 000. The results remain in accordance with the time scales ratio between solid and fluid states reported by He and Oldfield (2011). For quantitative comparison purposes, the RRMS deviation for the first 1000s of pre-warming process is provided in Figure 11. The TFEA...
methods, in addition to the ET simulations with SF 1000 and 10 000, achieve RRMS deviations of less than 3% and, for this reason, are preferred for numerical modelling of pre-warming processes with hot air. With regards to calculation times (cp. Table 1), the ET calculation with SF 1000 TS 0.001s requires 4.7 times higher computational effort than the FF method and, therefore, is not preferable for extensive simulation matrices. In contrast, with regards to quasi-constant heat fluxes through boundary interfaces in warm-keeping operations, the higher values of SF and TS enabling additional savings in computational time may be applied to the setup of the numerical model (even up to SF 10 000).

5 CONCLUSIONS

In this paper, four different numerical methods belonging to coupled and uncoupled simulation approaches are investigated in pursuit of the calculation of transient temperature fields in a pre-warmed steam turbine. The uncoupled TFEA-LINEAR and TFEA-EXPO calculations achieve good accuracy in comparison to the reference FF method (RRMS deviation under 2%), and enable a reduction in computational time by a factor of about 6. The accuracy of the coupled ET method significantly depends on the setup of the simulation. The calculation with SF 1000 and TS 0.001s provides the highest quality solution, although this requires 4.7 times higher computational resources. The additional savings in calculation times (factor 1.26) and good accuracy (RRMS deviation under 3%) are obtained by means of ET calculation with SF 10 000 and TS 0.001s. Therefore, this setup may be used to numerically solve the calculation matrix consisting of several operating points. To conclude, the numerical methods discussed in the presented work enable determination of transient temperature fields in turbine components in long pre-warming processes. As a result, the industrial partner is able to further optimize the pre-warming arrangement with regards to thermomechanical fatigue, in pursuit of faster turbine start-ups. In a future paper, a comparison of pre-warming simulation, based on one of the presented numerical methods, with experimental data will be performed.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Indices</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>t_∞</td>
<td>Time point before the start of a thermal shock process</td>
</tr>
<tr>
<td>t_|∞</td>
<td>Time point at the end of a thermal shock</td>
</tr>
<tr>
<td>av</td>
<td>Averaged</td>
</tr>
<tr>
<td>FL</td>
<td>Fluid</td>
</tr>
<tr>
<td>MP</td>
<td>Measuring point turbine wheel</td>
</tr>
<tr>
<td>ref</td>
<td>Reference</td>
</tr>
<tr>
<td>SL</td>
<td>Solid</td>
</tr>
</tbody>
</table>

Abbreviations

- CFD: Computational fluid dynamics
- CHT: Conjugate Heat Transfer
- ET: Equalized timescales
- FEA: Finite element analysis
- HTC: Heat transfer coefficient
- OP: Operating point
- RRMS: Relative root mean square

ACKNOWLEDGMENTS

The present work is a part of the joint research program COOREFLEX-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.

REFERENCES

- Vogt J., Schaaf T., Mohr W. F. D., and Helbig K. (2013). Flexibility improvement of the steam turbine of Conventional or CCPP.