Optimized Approach for Determination of the Solid Temperature in a Steam Turbine in Warm-Keeping-Operation
Dennis Toebben, Piotr Luczynski, Manfred Wirsum, Wolfgang Mohr, Klaus Helbig

To cite this version:
Dennis Toebben, Piotr Luczynski, Manfred Wirsum, Wolfgang Mohr, Klaus Helbig. Optimized Approach for Determination of the Solid Temperature in a Steam Turbine in Warm-Keeping-Operation. 17th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC2017), Dec 2017, Maui, United States. hal-02392302

HAL Id: hal-02392302
https://hal.archives-ouvertes.fr/hal-02392302
Submitted on 3 Dec 2019

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L’archive ouverte pluridisciplinaire HAL, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d’enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.
INTRODUCTION

Today, power plants using steam turbines are responsible for more than 50% of the worldwide power generation [1]. Most of these power plants have an age of thirty years and more [2]. Within the last decade, renewable energies are continuously gaining influence in the energy supply sector. These renewables lead to a fluctuating power feed-in into the grid which needs to be regulated by the conventional generation system. Most of the installed capacities of steam turbine power plants are designed for basic load with a limited ability for high load ramps and fast start-ups. Consequently, the existing power plants need to be revised and improved to increase the operational flexibility.

Further decreasing operating hours of the conventional power plants and thus an increasing cost pressure in the energy sector are another result of the growing share of renewable energies. For the decision whether a power plant goes on stream, the importance of the start-up costs is still increasing. The base load efficiency is no longer critical, but rather the integral efficiency including the start-up and shutdown procedure as well as base, part and minimum load operation.

One option to improve the flexibility and the integral efficiency is to keep the steam turbine in a warm condition between a shut-down and the next start-up. The required compensation of the losses can be achieved by the use of heat blankets [3], gland steam [4] or even by hot air, which is, for example, passed through the steam turbine [5]. The last approach is the basis for the present investigations.

For an efficient, fast and lifetime conserving start-up the knowledge of the temperature distribution within the thick walled components of the steam turbine is crucial. Especially the temperature of the rotor is a determining factor which is not measured in commercially operated steam turbines. The determination of the rotor temperature distribution can only be conducted with prohibitive expense. Therefore, a simplified calculation model is required.

Considerable improvements of the computing power and numerical simulation tools enable a relatively detailed investigation of the heat transfer and the flow phenomena in steam turbines [9]. However, detailed fluid-dynamic calculations can only be conducted for a limited number of numerical nodes so far. This leads to either smaller geometries or to coarser numerical meshes and consequently to a reduced accuracy. A highly accurate coupled fluid and solid 3D-simulation of a whole multistage steam turbine can only be conducted with prohibitive expense. Therefore, a simplified calculation model is required.

Several analytical approaches for calculating the heat transfer within a steam turbine are known from literature. Brilliant et al. [10] developed a separate calculation model for both an impulse and a reaction steam turbine, using differ-
A further approach was developed by Pusch et al. [12]. The aim of the investigation was to simulate the start-up procedure of an intermediate pressure steam turbine to analyze low-cycle-fatigue. Therefore, a numerical FEM model was developed including the rotor and the inner casing. Various analytical heat transfer correlations were used to calculate the heat transfer. The absorption of heat by the blades and vanes is calculated by a modified correlation which converts the HTC at the blade surfaces (Nusselt-approach developed by Traupel et al. [13]) to an equivalent HTC at the endwall. Therefore, only the blade and vane root needs to be integrated into the model.

Moroz et al. [14] conducted a transient thermo-structural analysis of a combined high pressure / intermediate pressure steam turbine during a start-up. For these investigations a FEM model was developed, which includes the rotor and the inner casing of the steam turbine. The FEM model was separated into several heat transfer zones in which analytical heat transfer correlations from literature were used. In addition to the convective heat transfer, the model is able to calculate condensation within the individual zones. Based on these analytical equations, the temperature distribution of the rotor was analyzed. These calculations were used to determine the rotor displacement in several discrete time steps during the start-up phase.

Wang et al. [15] investigate the impact of the steam temperature fluctuation during steady-state operation on the temperature field within the rotor. The heat transfer between the steam and the rotor is calculated by analytical equations concerning the rotor surface, the seal structure and the blade groove. The heat transfer coefficients serve as an input parameter for a FEM rotor model. It is assumed that all heat fluxes received by the blades are conducted into the rotor through the contact between blade root and blade groove. Similar to the approach of Pusch et al. [12], the blades are simplified as the blade roots and integrated into the model. In addition, an analytical approach for determining the contact resistance at the blade-rotor-connection depending on the rotational speed is included.

Born et al. [16] developed a numerical model of an intermediate pressure steam turbine for the investigation of steady state operation and natural cooling effects. The model includes the inner and outer casing and the labyrinth seals. The rotor is not modeled. The authors used several analytical heat transfer correlations based on approaches from literature for determining the heat transfer by forced and free convection, radiation and conduction.

All these investigations were made with steam as operational fluid. The present paper deals with an optimized approach for the determination of the solid temperature in a steam turbine during warm-keeping-operation with hot air. Hence, the previous calculation approaches are inappropriate. A simplified Hybrid-FEM model (HFEM) has been developed, which uses analytical heat transfer correlations coupled with a FEM model for time-optimized calculation of the solid body temperature distribution. The model consists out of a repetitive stage including inner casing, rotor, vane, blade and shrouds. In comparison to the approaches known from literature, the present model uses detailed numerical Conjugate-Heat-Transfer (CHT) investigations [9] to develop suitable heat transfer correlations for the most relevant surfaces concerning the heat transfer. The FEM model was developed on basis of the CHT model grid. The fluid grid was removed and the solid grid was coarsened to optimize the calculation time. Thus, on the one hand heat transfer and temperature analysis can be made with minimized calculation effort and on the other hand high resolution.

In the first part of the present paper, the model setup and the boundary conditions of the warm-keeping operation with air are introduced. Subsequently, the analytical calculation models for convection and radiation are presented. Finally, a comparison between the results of the numerical CHT model and the Hybrid-FEM model is conducted, which shows the advantages of the simplified model.

**NOMENCLATURE**

**Symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Area</td>
</tr>
<tr>
<td>a, b, b₂</td>
<td>Empirical parameter</td>
</tr>
<tr>
<td>F</td>
<td>View factor</td>
</tr>
<tr>
<td>h</td>
<td>Heat transfer coefficient</td>
</tr>
<tr>
<td>l</td>
<td>Characteristic length</td>
</tr>
<tr>
<td>m</td>
<td>mass flow rate</td>
</tr>
<tr>
<td>n</td>
<td>rotational speed</td>
</tr>
<tr>
<td>n</td>
<td>Number of operating points</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt-number</td>
</tr>
<tr>
<td>p</td>
<td>Pressure</td>
</tr>
<tr>
<td>Pr</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>Q</td>
<td>Heat flux</td>
</tr>
<tr>
<td>q</td>
<td>Area related heat flux</td>
</tr>
<tr>
<td>r</td>
<td>Radius</td>
</tr>
<tr>
<td>r₁ / r₂</td>
<td>Inner / outer flow channel radius</td>
</tr>
</tbody>
</table>
1. HYBRID-FEM MODEL

The *Hybrid-FEM model* uses hybrid calculation approaches: The heat transfer within the solid body is calculated by the use of a Finite Element approach while the heat exchange by forced convection, radiation or contact heat transfer is calculated by a semi empirical approach based on detailed Conjugate-Heat-Transfer (CHT) simulations.

Figure 1 provides detailed information about the model setup of the Hybrid-FEM model. In the case of fast and frequent start-ups and high load ramps, the impact on the lifetime is crucial. Therefore, information about the temperature and stress distribution of the turbine in such off-design operations are important. The Hybrid-FEM model can be applied to provide these information. A numerical fluid and solid mesh for one repetitive stage is generated from the basis turbine design. With this mesh, a CHT model is developed which is used for calculating a characteristic map regarding the operating range of interest (e.g. start-up phase, warm-keeping or pre-warming). To secure a appropriate resolution of the thermal boundary layer, the near wall heat transfer is modeled based on the Low-Reynolds-Number method (kω-SST turbulence model), which resolves the details of the boundary layer profile by using very small mesh length scales in the direction normal to the wall. The fluid boundary layer y+ is smaller than 1.0 to secure a fine mesh resolution. For transferring these calculation results into the Hybrid-FEM model, modified Nusselt-Correlations are used to fit the characteristic HTC map. In contrast to the requirements of the CHT model, a coarse solid body mesh is sufficient for the Hybrid-FEM model’s purposes. Hence, the fluid mesh is removed from the initial mesh and the solid mesh is coarsened. This repetitive stage mesh can be multiplied to any needed number of turbine stages (neglecting the flow channel expansion). In a final step the adapted analytical heat transfer correlations are integrated into the model. The result is the Hybrid-FEM model which is able to calculate the temperature and stress distribution within large scale turbine geometries with improved accuracy. This is reasoned by individually modified heat transfer equations, which provide a reduced as well as at least manageable calculation effort for simulating a whole multistage steam turbine.

In comparison to other approaches, the Hybrid-FEM model uses adapted instead of general heat transfer correlations for a specific turbine geometry and specific operation conditions. In the next step of the ongoing investigation the Hybrid-FEM model will be validated against experimental data. Due to the strong basis of the CHT simulations just a few regular operation measurements are sufficient to validate the model.

The presented Hybrid-FEM model is used to calculate a warm-keeping operation of a steam turbine. In general, the warm-keeping process is designed to increase the flexibility and reduce the lifetime consumption. The warm-keeping procedure is characterized by a hot gas (e.g. air) which is passed through the turbine to compensate the heat losses.
primarily to the ambient and the lubricant oil system. The steam turbine rotates during the warm-keeping process. Besides the conventional rotation direction, it is possible to rotate the steam turbine rotor in reverse direction to increase the warm-keeping efficiency [5]. The operating case with conventional rotation direction is called quadrant 1 (Q1) and the other one quadrant 2 (Q2).

The heat transfer and flow phenomena in warm-keeping operation have already been investigated [9]. For this analysis a numerical 3D single stage CHT-model was used characterized by the following assumptions:

- Steady-state CHT-simulation
- Single blade passage (approx. 2.5°) with periodic boundaries in circumferential direction
- Frozen rotor-stator interface
- Fully turbulent ideal gas with constant fluid properties
- No heat radiation
- Low Reynolds kω-SST turbulence model
- Constant material properties for the solid state
- Fluid boundary layer: $y^+ \leq 1.0$

Toebben et al. [9] figured out that in warm-keeping operation most of the heat is transferred by forced convection from the fluid to blades and vanes. These kind of fins conduct the heat through the blade-rotor respectively vane-casing connection to the main turbine components. Therefore, the contact heat resistance between these surfaces is important for calculating the overall heat transfer and the temperature distribution within rotor and casing. Besides convection, heat radiation within the flow channel and between the inner and outer casing occurs. The influence of the heat radiation depends on the surface temperatures. In contrast to the investigations of the flow phenomena and the convective heat transfer in which the radiative heat transfer could be neglected due to similar surface temperatures (steady-state operation), the relevance of radiation increases for transient operations. Thus, the three heat transfer phenomena convection, radiation and contact heat transfer need to be integrated into the Hybrid-FEM model by using analytical calculation approaches (Fig. 2).

The coarsened solid mesh of the single stage Hybrid-FEM model has only $5.1 \cdot 10^8$ nodes. In comparison to the solid mesh of the CHT model the reduction factor is 63. The quality and accuracy of the coarsened mesh was proven in a mesh study. The reduction of the number of nodes is permitted because there is no fluid mesh and thus, the interpolation between fluid and solid mesh is no longer crucial. The fluid mesh usually has a much higher number of nodes than the solid mesh to ensure a high resolution of the flow phenomena and the thermal boundary layer. The numerical solver needs to interpolate between the fluid and the solid mesh. Hence, the resolution of the solid mesh is also crucial for the accuracy of the heat transfer calculation. Once the fluid mesh is no longer needed, the solid mesh can be coarsened significantly. The total mesh reduction factor of the final Hybrid-FEM mesh compared to the fluid ($3.6 \cdot 10^6$ nodes) and solid ($3.2 \cdot 10^6$ nodes) CHT mesh is 133. With this mesh resolution it is possible to simulate a whole steam turbine with acceptable computational effort.

Before going into detail of the analytical calculation approaches, the relevant surfaces of the model should be introduced. The single stage model has eight main surfaces, which are responsible for the main heat exchange between fluid and solid as well as for heat radiation. These main surfaces are shown in Fig. 3.

2. ANALYTICAL APPROACH: CONVECTION

Each surface has an individual impact on the heat transfer from the fluid to solid domain. To quantify the sensitivity of the total heat flux on the main surfaces, the relative heat flux for each surface is illustrated in Fig. 4. For this comparison
a simulation with equal temperature on each surface was conducted. To consider the impact of different surface areas, both the absolute and the area related value of the normalized heat flux is given. The blade and the vane (including the tip) receive each about 48% of the heat in warm-keeping operation. This effect is enhanced by the large surfaces of blade and vane compared to the other surfaces, shown by the area related value. The remaining heat is absorbed by the shrouds of both, blade and vane. Whereas, the surfaces S1 Housing and R1 Rotor can be neglected. The separation of the blade surface into blade and blade tip serves to set separate fluid reference temperatures (due to the cooling effect of the shroud) and to include heat radiation between blade tip and blade shroud. For further evaluation only the four main surfaces Vane, Vane Shroud, Blade, Blade Shroud are considered.

For these four main surfaces an analytical approach for the convective heat transfer is developed. The numerically calculated heat transfer coefficients (HTC) \(h_{i,n}\) and Nusselt-number \(Nu_{CHT,i,n}\), which definitions are given in Eq.1 and Eq.2 serve as basis. The index \(i\) refers to the individual surface and the index \(n\) to the inlet plane of the flow channel section in which the regarded blade/vane row (S1/R1) is located. \(\dot{Q}_i\) is the area averaged heat flux of the surface \(i\) with the surface area \(A_i\). The reference temperature \(\bar{T}_n\) is the mass flow averaged temperature of the fluid passing the plane \(n\). \(\bar{T}_i\) is defined as the area averaged temperature of the surface \(i\).

\[
\bar{h}_{i,n} = \frac{\dot{Q}_i}{A_i(\bar{T}_n - \bar{T}_i)}
\]

\[
Nu_{CHT,i,n} = \frac{\bar{h}_{i,n} \cdot l_i}{\lambda_{fluid}}
\]

The heat transfer within the flow channel of the steam turbine in warm-keeping operation is mainly influenced by the inlet air mass flow \(\dot{m}\) and the rotational speed \(n\). Due to the assumption of atmospheric back pressure and very low mass flow rates, the inter stage pressure drop is very small and thus, the pressure influence is negligible. In order to describe the input parameters, the Reynolds-number in circumferential flow direction (Eq. 3) and in axial direction (Eq. 4) is defined. The radii \(r_1\) and \(r_2\) are the limits of the inner and outer wall of the flow channel. The characteristic length of the regarded surface is defined as \(l_i\).

\[
Re_{u,i} = \frac{\pi n (r_1 + r_2) l_i}{60 \nu}
\]

\[
Re_{ax,i} = \frac{ml_i}{\rho \pi (r_2^2 - r_1^2) \nu}
\]

For the development of the convective heat transfer correlation for the warm-keeping process in two different operations (Q1 & Q2) a physical correlation approach is used. The investigations have shown that \(Re_{ax} & Re_u\) are the dominating parameters for the description of the Nusselt-number. Therefore, a Nusselt-approach \(\overline{Nu}_i\) (Eq. 5) was chosen, including empirical factors \(a, b_1\) and \(b_2\), as well as the sum of both Reynolds-numbers and the Prandtl-number, to consider the fluid properties.

\[
\overline{Nu}_i = a_i (Re_{u,i}^{b_1} + Re_{ax,i}^{b_2}) Pr^{1/3}
\]

For each surface \(i\) and each operation condition (Q1 or Q2) one Nusselt-correlation is defined by individual empirical factors \(a_i, b_{1i}\) and \(b_{2i}\). These coefficients are determined by a minimization of the residual sum of squares based on the CHT results. The Nusselt-correlation of Eq. 5 is plotted in Fig. 5 for the conventional rotation direction (Q1) and in Fig. 6 for the reversed rotation direction (Q2). These figures also show the numerically calculated Nusselt-numbers for the convective heat transfer over the axial Reynolds-number with logarithmic scaled axes. In accordance to the findings of the flow analysis [9], the numerical results are influenced by secondary flow structures. In general, the Nusselt-number rises with increasing axial Reynolds-number and also with increasing circumferential Reynolds-number. Because of the single stage model structure, the vane is only influenced by the rotational speed when the axial Reynolds-number is extremely low. The blade shows a distinct dependency on the rotational speed. However, kinks in the Nusselt-number trend can be observed due to the secondary flow phenomena. In the vane and blade shroud, no distinct distribution of the calculated Nusselt-numbers can be found, especially for the reversed rotational direction (Q2).

The aim of the Nusselt-correlation is to reproduce the trend of the heat transfer based on physical relations. Thus, the correlation is not able to accurately describe the irregularities regarding changing vortex structures. To evaluate the accuracy of the correlation, the relative root mean square error \(rRMSE\) is introduced in Eq. 6, which represents the relative average deviation of the fitted response value \(\hat{y}(x_i)\) from the original data value \(y_i\).

\[
rRMSE = \sqrt{\frac{1}{n} \sum_{i=1}^{n} \left(\frac{y_i - \hat{y}(x_i)}{y_i}\right)^2}
\]
Optimized Approach for Determination of the Solid Temperature in a Steam Turbine in Warm-Keeping-Operation

Figure 5. Nusselt-number and convective heat transfer correlation - Q1

Figure 6. Nusselt-number and convective heat transfer correlation - Q2
The $r \text{RMSE}$ is given in Tab. 1 for each surface and the different operation conditions $Q_1$ and $Q_2$. The evaluation of the predicted values is made by the same CHT results, which are used for deriving the correlation. The $r \text{RMSE}$ of the $S_1$ Vane surface is less than ten percent. Due to the nonrotational inlet flow, the Nusselt-numbers of the vane show a distinct trend. The average $r \text{RMSE}$ is in a range of 14 to 18 percent. To consider the influence of the different surface areas, the area related average values of the $r \text{RMSE}$ are mentioned.

3. ANALYTICAL APPROACH: RADIATION

Besides the convective heat transfer, the heat radiation can also significantly influence the general heat exchange. Within the flow channel of a steam turbine, there are several complex geometries. The radiation heat exchange takes place between all infinitesimal surfaces. Hence, the calculation of these heat exchanges is very complex.

A few commercial numerical FEM software tools are able to calculate the radiative heat exchange without a fluid mesh. In the present work, the software ANSYS CFX 15.0 has been used, which does not provide this opportunity. The numerical CHT simulation (fluid & solid) of simultaneously convection and heat radiation reaches high radiation imbalances. Therefore, the heat radiation was not considered in the simulation of the convective heat transfer. However, to use the synergy of the CHT model and the Hybrid-FEM model, ANSYS CFX was chosen as software basis.

For the integration of the heat exchange by radiation into the Hybrid-FEM model, an analytical approach has to be found. In accordance with the literature [17] an approach for the radiative exchange between grey Lambert radiators is chosen. Each of the surfaces, which were defined before, exchanges radiative energy depending on the geometry of the surface and the position within the flow channel. To consider this effect, a view factor $F_{12}$ has to be found, which represents the share of radiative energy from a surface 1 to the surface 2. Equation 7 describes the commonly known analytical approach for heat radiation $Q_{rad}$ regarding the view factor and the emissivity $\epsilon_i$ of the related surface. In conservative estimation the emissivity is assumed as $\epsilon_i = 1$ for all surfaces. Due to the oxidation layer, the surfaces within the flow channel have a dark grey color.

$$Q_{rad,1\rightarrow2} = \frac{\sigma(T_1^4 - T_2^4)}{\frac{1}{\epsilon_1 A_1} + \frac{1}{\epsilon_i A_i} F_{12} + \frac{1}{\epsilon_2 A_2}}$$

For the determination of the view factors, the software tool ANSYS Fluent was used. Therefore, the numerical mesh was imported into Fluent. As calculation method Monte Carlo with the option surface to surface was used. Gas absorption respectively gas radiation is neglected. To consider also the share of radiation, which is exchanged between several stages, the single stage mesh was duplicated three times and connected to a simplified three stage model (without flow channel expansion). The surfaces of each stage were combined to surface groups as shown in Fig. 3.

In Fig. 7 the main surfaces and radiation heat fluxes are illustrated schematically. The blade (red) exchanges heat by radiation mainly with the neighboring blades in circumferential direction and the neighboring vanes in axial direction. The share of radiation energy, which is absorbed by other surfaces such as blades in the second row or parts of the rotor and housing, is less than ten percent. Besides the blade surface, the shroud surfaces (green) also exchanges radiation energy. The blade shroud collects a significant share of the radiation energy from the blade tip. Whereas, the shroud with its several fins absorbs most of the emitted energy by itself. For the vane (blue) it is similar. Thus, with two respectively four view factors (incoming & outgoing) for each of these main surfaces approximate 90% of the irradiation energy can be covered. In Fig. 7, the continuous arrows demonstrate these main heat exchanges. The arrows with dotted lines show the secondary exchanges.

For the verification of these calculated view factors with the solver ANSYS Fluent, the solver ANSYS CFX was used. The view factors were calculated separately. For this purpose, three simulations for the determination of each view factor were conducted (Monte Carlo, surface to surface). In the first simulation all other surfaces apart from that two for which the view factor should be determined were defined.
as "openings" with a surface temperature of zero Kelvin. As a result, the emitted and absorbed heat flux is calculated, including the share of irradiation which is absorbed by the emitting surface itself. Therefore, two further simulations were conducted to each calculate only the irradiation heat flux which is absorbed by the regarding surface itself. The average relative error of these two calculation methods is five percent. This value meets the expected accuracy.

4. MODEL COMPARISON

For the comparison of the HFEM and the CHT model, all operating points were recalculated with the HFEM model. Due to the high numerical imbalances in the energy equations while solving heat radiation, the radiation influence had to be deactivated for no distorting the results. In Fig. 8 and Fig. 9 the area related heat fluxes of the main surfaces are compared between the HFEM model on the x-axis and the CHT model on the y-axis. Figure 8 represents the first (Q1) and Fig. 9 the second quadrant (Q2). It can be seen that the results of the area related heat fluxes match in good agreement. For a quantitative analysis, the $rRMSE$ (similar to Eq. 6) of each surface is calculated and shown in Tab. 2. This error estimation takes the general error of the model into account, which is also influenced by the calculation of the fluid reference temperature and the area averaging, in contrast to the $rRMSE$ of the Nusselt-number (Tab. 1). It can be noticed that the $rRMSE$ of the heat fluxes is lower than that only of the Nusselt-number. The average area related $rRMSE$ which considers the different surfaces areas and thereby, the impact on the general heat transfer, is in between 0.1 and 0.13.

Table 2. $rRMSE$ of the area related heat flux

<table>
<thead>
<tr>
<th>Surface</th>
<th>Q1</th>
<th>Q2</th>
</tr>
</thead>
<tbody>
<tr>
<td>R1 Blade</td>
<td>0.0683</td>
<td>0.1407</td>
</tr>
<tr>
<td>R1 Blade Shroud</td>
<td>0.1729</td>
<td>0.2677</td>
</tr>
<tr>
<td>S1 Vane</td>
<td>0.0533</td>
<td>0.0691</td>
</tr>
<tr>
<td>S1 Vane Shroud</td>
<td>0.1819</td>
<td>0.1407</td>
</tr>
<tr>
<td>Average</td>
<td>0.1266</td>
<td>0.1546</td>
</tr>
<tr>
<td>Average (area related)</td>
<td>0.1027</td>
<td>0.1332</td>
</tr>
</tbody>
</table>

In Tab. 3 the calculation effort for a steady-state calculation of the different single stage turbine models is compared. The calculation time is provided in core-h, which represents the number of computing cores multiplied with the calculation time in hours. In contrast to the HFEM, the calculation effort for a CFD (fluid) and a CHT (fluid & solid) is shown. The calculation effort for a CFD simulation is factor 410 and for the CHT simulation factor 660 higher than the simplified HFEM model (without initial file). Thus, the HFEM calculation does not need large-capacity computers.

Table 3. Comparison of calculation effort (steady-state)

<table>
<thead>
<tr>
<th>Model</th>
<th>Core-h</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>HFEM</td>
<td>0.1</td>
<td>1</td>
</tr>
<tr>
<td>CHT</td>
<td>66</td>
<td>660</td>
</tr>
<tr>
<td>CFD</td>
<td>41</td>
<td>410</td>
</tr>
</tbody>
</table>

Figure 10 illustrates a temperature contour plot of the CHT model on the left and the HFEM model on the right side. The temperature distribution is nearly identical. The highest deviation is reached at the tip of the blade and vane, due to the highest relative error of the predicted heat transfer coefficients at the shroud surfaces.
were integrated into this simplified model. In contrast to other approaches, a detailed five taboldstyle. CONCLUSION to investigate steady-state warm-keeping as well as transient air. efficients and to adapt empirical factors of a heat transfer lyzed to develop a multistage steam turbine model. The aim is contact heat resistance (see Figure eight taboldstyle) will be integrated into the mesh and definition of relevant surface groups. The ana- so far.

turbine, for which the calculation effort has been exorbitant the warm-keeping with air - for a whole multistage steam step to calculate a characteristic map of heat transfer co- /eight taboldstyle/five taboldstyle % in less than zero taboldstyle/zero taboldstyle/zero taboldstyle times of the calculation time. This ap- results of detailed CHT simulation with a accuracy of above 95 % in less than 600 times of the calculation time. This approach enables the simulation off-design operation - such as the warm-keeping with air - for a whole multistage steam turbine, for which the calculation effort has been exorbitant so far.

In further investigations an analytical approach for the contact heat resistance (see [18]) will be integrated into the HFEM model. Furthermore, inter stage effects will be ana- lyzed to develop a multistage steam turbine model. The aim is to investigate steady-state warm-keeping as well as transient pre-warming processes for a whole steam turbine with hot air.

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 the German ministry of economy and technology under grant number 03ET7041B. The authors gratefully acknowledge GE Power for their support and permission to publish this paper.

REFERENCES


5. CONCLUSION

The operation of turbines under extraordinary conditions becomes more important in the context of flexibility. The numerical simulation of such operating conditions can help to understand the risks as well as advantages.

In this present paper, an optimized approach for the determination of the solid temperature in a steam turbine was introduced. In contrast to other approaches, a detailed Conjugate-Heat-Transfer (CHT) model was used in a first step to calculate a characteristic map of heat transfer co-efficients and to adapt empirical factors of a heat transfer correlation. In a next step, a simplified solid body model was created from the CHT model by coarsening the numerical mesh and definition of relevant surface groups. The analytical calculation approaches for convection and radiation were integrated into this simplified model. This model - so called Hybrid-FEM model (HFEM) - is able to reproduce the results of detailed CHT simulation with a accuracy of above 85 % in less than 600 times of the calculation time. This approach enables the simulation off-design operation - such as the warm-keeping with air - for a whole multistage steam turbine, for which the calculation effort has been exorbitant so far.

In further investigations an analytical approach for the contact heat resistance (see [18]) will be integrated into the HFEM model. Furthermore, inter stage effects will be ana- lyzed to develop a multistage steam turbine model. The aim is to investigate steady-state warm-keeping as well as transient pre-warming processes for a whole steam turbine with hot air.

Figure 10. Model comparison: Hybrid-FEM model vs. CHT model ($Re_{ax} = 1.2 \cdot 10^3, Re_u = 2.3 \cdot 10^4, Q_1$)


