Numerical Computation of Heat Transfer Coefficient for a Radial Compressor Stage using CFD and Comparison with Analytical Model Computations

Prof. Dr.-Ing. Marius Geller,
Dipl.-Ing. Norbert Kluck, Dipl.-Ing. Roland Magiera
University of Applied Sciences Dortmund, Dortmund, Germany

Introduction

CFD simulation is increasingly used in the turbo machine industry. Computation using Fluid-Structure Interaction (FSI) is currently standard technology. The effects of fluid on solid bodies can be determined with great precision both transiently and in steady state. An important field of application is the simulation of temperature distribution within a complete turbo machine. Precise forecasts of thermal stresses combined with high pressures can warn of potential high thermal and mechanical loads for the stages of a turbo machine, leading to improved machine efficiency. As time is a dominant factor, time-intensive FSI computations are rarely a realistic option in industrial development work. On their own, steady state and/or transient temperature simulations of structural regions are possible at a fraction of the time required for fluid-structure interactions if the HTC, as the most important boundary condition for steady-state or transient thermal simulations, is known.

Keywords

CFD, Computational Fluid Dynamics, HTC, Heat Transfer Coefficient ($\alpha$), mesh quality, Compressor with gap, Impeller without gap, straight channel, local and averaged equations from VDI

Aim

The aim of this investigation was to determine the HTC for all relevant regions in a radial turbo machine stage, and for which a modelled complete turbo machine stage with impeller, guide wheel and gap influence was simulated. HTCs computed by CFD were standardized by rendering them dimensionless using analytical equations from the VDI Wärmeatlas and in this context direct comparison with the empirical approach made.
1 Method

Four different phases were defined for the project. Starting from complete simulation of a compressor stage with its associated return channel and all gap leakage, the three models in the subsequent project phases represented increasingly simplified constructs (see Table 1). The results of all models were compared with each other and with those of empirical computations from the VDI Wärmeatlas.

<table>
<thead>
<tr>
<th>Phase</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>complete compressor with impeller, guide wheel and gap leakage</td>
</tr>
<tr>
<td>2</td>
<td>impeller and without gap leakage</td>
</tr>
<tr>
<td>3</td>
<td>straight rectangular channel</td>
</tr>
<tr>
<td>4</td>
<td>analytical Gnielinski equation according to for general channel flow</td>
</tr>
</tbody>
</table>

Table 1. The different models for each of the four phases.

At the start of the investigation a complete compressor stage with gap was simulated. Subsequently, the impeller channel derived from the compressor with gap was evaluated without gap. The investigation was concluded by simulating straight, stationary and straight rotated channels of the form that best fits the geometric formulae derived from the VDI Wärmeatlas. The models and their simulation results are detailed and explained in the following sections.

1.1 Compressor with gap

The compressor stage was modelled as completely parameterized. The individual components of the geometry are flexibly constructed such that any subsequent modifications can be done keeping time costs to a minimum. In addition, it was also made possible that external data, such as impeller blade measurements – be integrated into stage geometry so they can be replaced later by modified data.

Modelling started with a 2D meridian contour layout of compressor geometry (see Fig. 1). The complete modelling process is constructed such that these dimensional alterations are transferred fully parameterized to the 3D mode, shown in Fig. 2. Since the software used is also configured to manage the mesh parameterized, this in turn means it can also accommodate modifications to geometries at no great cost. Furthermore, it is possible using certain in-house software (Blade Runner) to replace mapped impeller blades with modified examples and for the 3D model to incorporate these alterations. When modelling it is particularly important to ensure that part volumes produced during partitioning consist of regular six-plane part volumes as only these are permitted when meshing with structured hexahedral elements (mapped mesh; see Fig 3). FSI simulations require extremely high-quality meshing in the boundary layer region when the high precision calculations of heat transfer coefficient are required. In order to reduce the computer working time and storage capacity required, the following procedure was followed when running simulations: both a very coarse and very fine mesh was created for each flow region. The coarse mesh served for initialization and global definition of flow status in each region. The fine mesh was used for regions where local determination of heat transfer coefficient was required. The principle dimensions of the compressor with gap, including operating data are listed in Table 2.

| Impeller blade, outer | 400mm |
| Collar dia. | 100mm |
| Suction port dia. | 210mm |
| Return channel, outer | 620mm |
| Revs. per min. (rpm) | 13800 1/min |
| Intake temp. | 16°C |
| Intake pressure | 1bar |

Table 2. Principle dimensions and boundary conditions of the computer model
Adherence to certain conditions concerning mesh quality is necessary when investigating other geometries and these are described in detail in section 4. These conditions for mesh quality have been incorporated in meshing impeller sections. Figures 4-6 illustrate boundary layer resolution for impeller channel in detail.

1.2 Impeller without gap

Evaluation of the compressor stage with gap revealed that the gap influenced thermo-dynamic processes in the impeller and therefore comparisons with simplified equations from the VDI Wärmeatlas were less than convincing. For this reason it was decided during the project to simplify the geometry of the Phase 1 construct. Since HTC's in the impeller channel were the point of focus, the latter was modelled for a complete range of inflow and outflow regions and all the surrounding structural areas.

Figure 7 shows the model. The accompanying mesh is shown in Figure 8.
1.3 Straight channel

The straight channel represents the greatest degree of simplification in this procedure. It principally serves for comparison of ANSYS CFX 10 flow simulations with empirical data from the VDI Wärmeatlas. This investigation required a geometry that met the requirements of the VDI Wärmeatlas and understandably, was orientated to the principle measurements of the impeller. The version comprising a non-rotating channel with constant temperature at the structural surface fully complied with the conditions required by the VDI Wärmeatlas. Accordingly, it served as check of comparability between the computational approaches. The rotating version of a straight channel referenced to analytical equations was expected to reveal possible rotational effects. The principle measurements and operating data of the straight channel are listed in Table 3. Figure 9 shows the geometry and mesh of the straight channel.

<table>
<thead>
<tr>
<th>Principle dimensions of the straight channel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outer diameter</td>
</tr>
<tr>
<td>Inner diameter</td>
</tr>
<tr>
<td>Channel width (square)</td>
</tr>
<tr>
<td>Wall thickness of structural region</td>
</tr>
<tr>
<td>Length of structural region</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Boundary conditions for flow simulation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Revs. per min.</td>
</tr>
<tr>
<td>Intake temp.</td>
</tr>
<tr>
<td>Wall temp.</td>
</tr>
<tr>
<td>Intake pressure</td>
</tr>
<tr>
<td>Mass flow (4 cases)</td>
</tr>
<tr>
<td>Turbulence model</td>
</tr>
</tbody>
</table>

Table 3. Principle dimensions and operating data of simulation model for straight channel

The straight channel was measured by CFD for both the stationary and rotating state to determine possible effects of rotation on heat transfer.

1.4 Local and averaged equations from VDI

The analytical computations were made using the generally applicable Gnielinski equations for full turbulent channel flow taken from the VDI Wärmeatlas. Equation 1 (Eq. 1) was used to determine the Nusselt number and Equation 1b the local Nusselt number. The coefficient of friction was calculated from Equation 2 for both cases.

\[
N_u = \frac{(\xi/8) \cdot Re \cdot Pr}{1 + 12.7 \cdot \sqrt{\xi/8 \cdot (Pr^{2/3} - 1)}} \cdot (1 + \left(\frac{D_L}{L}\right)^{2/3}) \quad \text{(Eq. 1)}
\]

\[
N_u, \text{ local} = \frac{(\xi/8) \cdot Re \cdot Pr}{1 + 12.7 \cdot \sqrt{\xi/8 \cdot (Pr^{2/3} - 1)}} \cdot (1 + 1/3\left(\frac{D_L}{x}\right)^{2/3}) \quad \text{(Eq. 1b)}
\]

\[
\xi = (1.8 \cdot \log(Re) - 1.5)^2 \quad \text{(Eq. 2)}
\]

The heat transfer coefficient was determined with hydraulic diameter as:

\[
\alpha = \frac{N_u \cdot \lambda}{D_h} \quad \text{with} \quad D_h = \frac{4 \cdot A}{U} \quad \text{(Eq. 3 and Eq. 4)}
\]
2 Evaluation of mesh quality

Before computing heat transfers in impellers, comprehensive testing of the required mesh fineness was first carried out. The straight, non-rotating channel is provided and simulated with meshes of varying degrees of fineness and compression (BIAS). The fineness of mesh used is illustrated in Figs. 10-13, including \( y^+ \) values. In order to correctly record the effects due to so-called thermal attempt, the mesh is compressed at both the leading and trailing edges (Fig. 14).

Figure 15 shows the results of CFD simulation. The histogram shows the average \( \alpha \) number determined for total channel length for the following cases:

- channel without BIAS (i.e. mesh compression at the site of so-called thermal attempt)
- channel with BIAS
- analytically according VDI Wärmeatlas (Equation 1)

Good agreement is apparent between simulation results from CFD and analytical calculations for the fine mesh with a \( y^+ \) value of 1.4 and BIAS for the leading and trailing edges. Due to the distribution of columns, which indicate a rise in divergence between simulation and analytical results for \( y^+ \) values less than 1, one can propose that \( y^+ \) values around 1 provide the best result.

Figure 15. Histogram of mesh comparisons

3 Evaluation

The computed values for heat transfer coefficients (\( \alpha \)) for all phases (=models), as described in section 3 were returned. Since local heat transfer coefficients cannot be directly calculated in CFD, they were calculated as follows:

\[
\alpha = \frac{q}{T_w - T_\infty} \quad (\text{Eq. 5})
\]

Values for heat flux \( q \), temperature at the wall \( T_w \) and in the fluid \( T_\infty \) were extracted as points from previously defined, taking care that the polylines for heat flux \( q \), wall temperature \( T_w \) and fluid \( T_\infty \) lie outside the boundary layer. Data was exported from CFD and evaluated by external software. The location of the polylines for the straight and impeller channels are shown in Figs. 16 and 17 respectively.
3.1 Impeller channel with gap

Plots of standardized heat transfer coefficients were relatively horizontal for the hub and shroud regions (see Fig. 18). Standardized values for the hub were somewhat higher than those for the shroud. The thermal attempt was absent from both plots. The reason was evaluation lines started in the region of the cut back leading edge, whereas the heat transfer naturally had its effect at the leading edge of the impeller. Approximately 90% of the channel lengths showed a surge for the shroud plot. This is can be attributed to the shroud gap. This resulted in turn from the difference between the fluid and structure temperatures is zero since heat in the structural region could not dissipate away.

The temperature distribution shown in Fig. 20 highlights the strong influence of the gap on thermo-dynamic behaviour within the stage. It is clear to see that the temperatures of all the parts perfused, such as the return channel and hub and shroud on the impeller outlet, were clearly higher than that of the fluid. This is attributed to frictional effect of flow on the wall and the fact that heat cannot dissipate away thereby giving rise to overheating of the solid structures. In turn, this resulted in the definition of the numerator in Eq. 5 being zero, explaining the aforementioned upswing in the plots. Values from CFD simulation compared to analytical results deviated by 0% to 30% and from 10% to 50% for the hub and shroud respectively. Plots for the pressure and suction sides (Fig. 19) were fundamentally different in comparison to the plots for the hub and shroud. Firstly, a clearly defined thermal attempt was present in the initial 20% of the impeller channel. Secondly the differences in heat transfer coefficients between the suction and pressure sides were significantly different. Thirdly, the plots displayed diametrically opposite trends in the latter third of the impeller channel: heat transfer coefficient dropped sharply on the suction side whereas on the pressure side it rose sharply. Both are the result of different fluid velocities in the blade outlet /trailing edge of the channel.
Pressure side CFD simulation values for heat transfer coefficient lay approximately between 30% above and 15% below the analytical values. On the suction side, standardized heat transfer coefficient values deviated from analytical values by approximately 110%.

3.2 Impeller channel without gap

The impeller model without gap showed a relatively constant plot for standardized heat transfer coefficients for hub and shroud (Fig. 21). Variance was approximately 10% to 35% from the analytical values. As an equivalent to an impeller with shroud gap, analytical values were generally higher. A thermal attempt was also absent, as described in section 3. The shroud plot in this case had a somewhat larger divergence from as that for the hub plot, between 0% and 45% compared to the analytical computations.

![Figure 21: Standardised HTC on hub and shroud](image1)

A somewhat moderate thermal attempt occurred in the first 10% of the impeller channel on the suction and pressure sides of the impeller (Fig. 22). After a relatively brief plateau for both plots, suction side heat transfer rose sharply and then fell off in the last third of the channel due to fluid velocity at the trailing edge of the blade. In contrast to hub and shroud plots for the impeller with gap, these plots showed no dramatic upswing. A possible explanation can be found in the temperature distribution in Figure 23 and the boundary conditions of the equation. The outer area of the hub and shroud is provided with a constant temperature for the computations intended in this section, such that overheating of the structure, as with the gap model, does not occur, as one can clearly see in Fig. 23. Fluid and structure temperatures are shown with different scaling, in order to meaningfully colour differentiate the lower temperatures on the entire surface of the structure from those of the fluid.

![Figure 22: Standardised HTC at the blade of hub and shroud](image2)

The plot of standardized heat transfer coefficients on the pressure side was the complete opposite of that for the suction side. At first the values dropped slightly but then increased sharply towards the end of the channel. Even on this side, as with suction side, fluid velocity at the impeller outlet had a powerful effect. In summary, standardized heat transfer coefficients on the pressure and suction sides expressed a general tendency to higher deviation than on the hub and shroud region, and varied between 0% and 80%.

![Figure 23: Temperature distribution of impeller without gap](image3)

![Figure 24: Velocity distribution of impeller section without gap](image4)
3.3 Straight rotating channel

The standardized heat transfer coefficients for straight rotating channels show classical plots. After a thermal attempt in the first third of the impeller channel, during which the deviation from analytical computations dropped from 30% to 10%, the curves fell further to a constant deviation of 30%. The results of the pressure side and rear were generally better than those from the suction side. All three curves share a common feature, namely that analytical values were consistently high than those from CFD simulations.

![Figure 25. Standardized HTC on pressure side](image1)

![Figure 26. Standardized HTC on suction side](image2)

![Figure 27. Standardized HTC at rear](image3)

![Figure 28. HTC at diff. Reynolds Number](image4)

However, the behaviour of the curves relative to the maximum values of all heat transfer coefficients against Reynolds number agreed with expectation. In Fig. 28 one can clearly recognize higher heat transfer in the thermal attempt in the straight channel. The curves for different Reynolds numbers fell sharply here. The channel length over which the sharpest fall was observed, and during which the thermal attempt has an effect, ranged from 25% for a Reynolds No. of 40,000 to 40% for a Reynolds No. of 75,000. Heat transfer was relatively even across the remainder of the impeller channel. The standardized heat transfer values displayed in Fig. 27 indicate that analytical equations from the VDI Wärmeatlas exhibit an average deviation of approximately 20% from the CFD computations. Within the thermal attempt, deviation was approximately 5%. Standardization of a large part of the thermal attempt results in deceptively steep fall in the curves.

3.4 The straight, stationary channel

Evaluation of the straight, stationary channel produced results similar to those for the rotating channel. No difference was observed between the suction and pressure side, such that the results can be displayed in the following two diagrams.
4 Comparisons

4.1 Comparison of impeller channels with and without gap

Comparison of the curves for impeller models with and without gap showed a further point of agreement. Curves for the pressure side were almost identical, only diverging in the latter third of the plot. The value for the impeller with gap at 45% variance from the analytical computation was somewhat better than that for the impeller without gap, where deviation from analytical computations reached up to 100%. Curves from the suction side deviated from one another in the mid region. The model with the gap rose in contrast to that without gap. This rise might be expected since a pressure rise inside the system occurs and thus corresponding rise in heat transfer coefficient.

In the model with gap a possible rise in standardized heat transfer coefficient is compensated by the effect of the shroud gap.

Comparison of the lots of hub and shroud for both models reveals the excellent agreement between standardized heat transfer coefficients.
The plot of the model without gap is somewhat more even than that with gap. The upswing in the gap model reveals the complex effects of the gap, with Figs. 33 and 34 illustrating examples of velocity distribution.

Figures 35 and 36 show relatively good agreement between standardized values for hub and shroud regions in impeller sections both, with or without a gap. The essential difference between the two models is revealed in the latter 30% of channel length. The curves that express standardized values at the impeller disk fell off in the impeller section with gap, whereas standardized values for the impeller section without gap remained relatively constant. The plots for standardized heat transfer coefficients at the shroud differed significantly between both models. In the impeller section with gap, the plot was relatively constant, although towards the end deviation from analytical values was less than for the hub. In contrast, the plot of the model with gap showed a clear upswing at the end, as previously explained in section 5.1.

4.2 Comparison of impeller channel with gap and straight rotating channel

The general trend of plots for model impellers with gap and rotating straight channel showed good agreement. Both models had a clearly defined thermal attempt in proximal part of the impeller channel; in the distal part there was a tendency to drop off slightly. In the last third, a clear difference between the two models became apparent. The strongly uneven flow behaviour at the suction and pressure side of the model with blades, absent in the straight channel model, led to strong gradients of standardized heat transfer coefficients at the impeller outlet. Comparison of absolute differences between standardized values showed clear divergence, especially at the suction side for rotating straight channels.
was not present. Thirdly, impeller models with gap at the outlet showed a fall off of plot at the hub, where the gap has its influence. Comparison of standardized heat transfer coefficients reveals that the differences were relatively small. In both cases, analytical values lie above those derived from the CFD simulation.

4.3 Comparison of impeller channel without gap and straight, rotating channel

Generally, the trend described in the previous section also applies to comparison of impeller with gap and rotating, straight channel, resulting in much the same picture. Comparison between suction and pressure sides revealed a difference only in the mid-region of the suction side, where a rise in standardized heat transfer coefficient occurred, explained by the increased pressure in the impeller.
4.4 Comparison between straight stationary and rotating channels

Comparison between the straight stationary and rotating channels show an almost identical plot. This indicated a comparable small effect of rotation on heat transfer at suction and pressure sides.

Figure 45: Standardized HTC at rear of straight rotating channel

Figure 46: Standardized HTC at rear of straight stationary channel

Inspection of HTCs with respect to maximum values revealed slight differences. The value of the rotating channel plot was somewhat higher than that for the stationary channel.

Figure 47: Comparison of HTC from straight rotating and stationary channels

Figure 48: Comparison of HTC suction and pressure sides in a straight rotating channel

Any difference between the plots for pressure and suction side was generally very small.
5 Summary

The following conclusions may be drawn from this series of investigations:

- Local CFD simulated HTCs for a straight channel are generally 20% lower than results computed from analytical VDI equations.
- Comparison between CFD simulations in a stationary and rotating channels indicate that the rotational effect has an effect of approximately 10% on local HTC.
- The findings of CFD HTC simulation for a simplified straight channel were not transferable to a rotating impeller: differences of 110% were observed.
- Comparison between two impellers, with and without gap influence, indicated that analytical equations are not suitable for pre-emptively working out local HTCs in flow channels.
- The obligatory boundary conditions prescribed by VDI equations – constant heat flux or constant wall temperature – do not exist for impellers with gaps.
- In contrast to straight channels, the rotation effect at the suction and pressure sides of impellers results in deviation of up to 100%. This is the case with impeller simulations with or without taking gap influence into consideration.

6 Conclusion

Other results not published here, from analyses of HTC in other machine geometries, in particular gap to hub and shroud, and labyrinth, confirm that simplified analytical models deliver erroneous results on local heat transfer coefficients. Further investigations with complex geometries and analyses are necessary.