COMBINED OPTIMISATION OF THE LAST STAGE AND DIFFUSER IN A STEAM TURBINE USING META-MODEL

Ch. Musch[1] - *K. Cremanns*[2] - *S. Hecker*[1] - *A. Penkner*[1]

[1]Siemens AG
Rheinstraße 100
45478 Mülheim an der Ruhr, Germany
christian.musch@siemens.com

[2]Niederrhein University of Applied Sciences
Reinarzstraße 49
47805 Krefeld, Germany
K.Cremanns@hs-niederrhein.de

ABSTRACT

In this study a strategy for a 3D optimisation of the exhaust of a low pressure (LP) steam turbine is presented. The flow domain utilized consists of both the last stage and the exhaust diffuser. The optimisation is done with the help of a hybrid surrogate model for the diffuser flow. In the first part of the paper a numerical model and its validation is presented, which allows for a precise simulation of the diffuser flow including the actual 3D geometry. The second part describes the optimisation procedure and the necessary simplifications applied to this model in order to get a numerical setup that is fast enough to actually perform a design study with roughly 200 design variants within a feasible time. This model is used afterwards to create a surrogate model. Based on this meta model an optimisation is carried out and finally scrutinzed with a flow simulation of the whole exhaust hood.

KEYWORDS

STEAM TURBINE, EXHAUST, DIFFUSER, OPTIMISATION, META-MODELS

INTRODUCTION

Even with a somehow shifted focus in the design of modern steam turbines towards higher flexibility, the efficiency is still a major driver in the design process. Therefore, the optimisation of the cold end of the LP turbine remains a large contributor to raise the overall efficiency of the steam expansion process. The goal of such an optimisation will always include the minimisation of leaving loss, as the kinetic energy contained in the exit velocity of the diffusor cannot be converted into shaft work and thus lowers the efficiency of the turbine. Investigations by many authors have shown that the interaction between last stage and exhaust is very complex and has a large influence on the perfomance of both parts. To account for all the effects of the large radial extent of the last stage and the asymmetric flow in the exhaust hood, a 3-dimensional (3D) investigation of the flow including the complete row of the last stage running blade is considered state of the art within the design process. However, a numerical optimisation based on this approach is still outside the possibilities within the design process. Optimisation strategies presented in the past have often the disadvantage of neglecting the asymmetry of the exhaust flow. The procedure presented in this paper shows a possible way to account for the major influences of the flow and still be able to create an optimised 3D diffuser shape.

Prior Art

In order to improve the exhaust design, experimental (e.g. Finzel et al. (2011) or Taylor et al. (2016)) and in recent years especially numerical studies (e.g. Wang et al. (2010), Verstraete et al.

(2012) or Yin et al. (2016)) have been carried out. With an increase in computational capacity it was possible to actually carry out these studies using 3D CFD . However, the calculations are still very time consuming if the complete exhaust hood is taken into account. Therefore, the mentioned studies did not include the turbine blading. However, in many previous studies it has been pointed out by the authors that the inflow conditions to the diffuser are vital for a reliable prediction of the exhaust hood performance. The swirl (e.g. McDonald et al. (1971) or Fu and Liu (2010)), tip leakage jet (e.g. Nicoll and Ramparian (1970) or Finzel et al. (2011)) and total pressure (e.g. Hirschmann et al. (2010)) are thought to be the main influencing factors of the turbine on the diffuser flow. In recent years it has become popular to not only simulate the flow field in the exhaust hood separately, but rather include the last stage in the simulation to fully grasp the interaction between both (e.g. Stein et al. (2015), Burton et al. (2013), Li et al. (2012) or Polklas (2004)). While these models offer good results with respect to the exhaust performance, a numerical optimisation using these whole exhaust models is still not feasible within the design process. Therefore, throughflow solvers are still an option to be used in an optimisation (e.g. Musch et al. (2013) and Taylor et al. (2016)). The method presented in this paper is quasi 2-dimensional, but uses a combined model of the exhaust diffuser and last stage.

NUMERICAL MODEL PART 1 - VALIDATION OF WHOLE EXHAUST HOOD CFD

In this section a numerical model is presented that allows for an accurate prediction of the complete flow field in the exhaust hood. The numerical model is validated with measurements gained from a model turbine.

Model Turbine

The design of the model turbine has been described in great detail by Völker (2006). An image of the open turbine and a cross sectional view can be found in figure 1. The turbine is

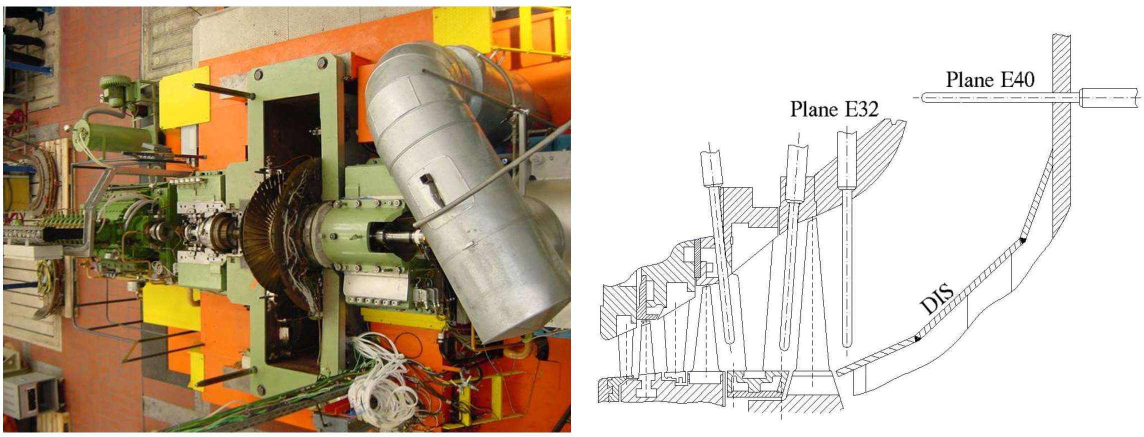

Figure 1: **Image and longitudinal section of model turbine**

a scaled down test rig which is operated with steam. Steam pressure and wetness are comparable to a real turbine. The scaling factor of roughly 4 results in a rotational speed of $n = 12600\ \mathrm{min}^{-1}$. The last three stages of a low pressure turbine are investigated. The last stage blade hub-to-tip ratio is about 0.45 and the exit Mach number is Ma ≈ 0.6. The turbine is equipped with various measuring devices, but for the purposes of this paper we will focus on the data gained from the probes in the diffuser. As for this study the focus is laid on the exhaust flow, the flow traverse measurements in plane E32 and plane E40 are used for the validation of the numerical model. In plane E32 the radial flow profiles have been collected at two different positions around the circumference (above and below the half joint). Plane E40 is only measured at one position at the top. The exact positions are shown in the corresponding figures of the measurements. The flow field in the exhaust hood is highly asymmetric due to the geometry of the exhaust hood. Therefore, the differences in the flow profiles between both positions in plane E32 already give a good indication whether the numerical model is capable to correctly predict the flow in the exhaust hood. As the measurement at the diffuser exit is at a third circumferential position (i.e. 12 o'clock), the complete asymmetry of the flow is covered by the measurements. Additionally, wall pressure measurements at the diffuser inner shell (DIS) were carried out at various positions around the circumference.

Numerical Model

The simulation approach chosen for the investigation presented here is similar to the one described by Stein et al. (2015). Contrary to the approach by Stein in this paper it has been decided to model all rotor passages. This way a more accurate representation of the flow asymmetry can be expected. With all rotor passages being modeled in one domain, no cyclic periodicity has to be applied in the rotor domain. As last stages of the size investigated in this paper are almost completely choked in the last stage rotor blade, circumferential variations at the stator vane are small. Considering this and with respect to computational time only one stator passage has been modeled. Consequently, the numerical model (as shown in figure 2) consists of a single passage of the last stage stator vane and the complete row of the last stage rotor blade, which are coupled to the exhaust hood domain via multiple mixing planes, i.e. each rotor passage has a corresponding section at the diffuser inlet plane. Hence, the number of mixing plane interfaces is equal to the last stage blade count. A typical interface of a rotor passage outflow and the corresponding diffuser inlet section is also shown in figure 2. The stator vane is coupled to the rotor blade domain via a mixing plane as well. Boundary conditions at the inflow of the stator

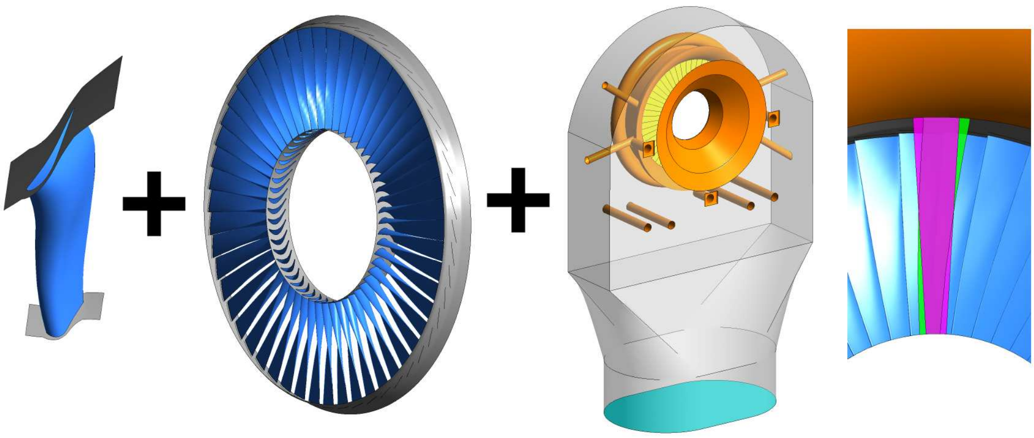

Figure 2: **Setup of the numerical model for the whole exhaust CFD**

vane are the stagnation pressure and temperature as determined by through-flow calculations. Additionally, the flow directions have been prescribed. At the exhaust hood outlet an average static pressure corresponding to the condenser pressure has been prescribed. Steam is modeled as an ideal gas in this setup. The heat capacity at constant pressure is set to 7365 $\frac{J}{kgK}$ and the heat capacity ratio to $\gamma = 1.063$. As described by Havakechian and Denton (2015) the error compared to a CFD with wet steam is small enough for the purposes of this paper. Only if a more detailed look on the performance regarding wetness losses is desired the fluid needs to be modeled from a steam table. Turbulence closure is achieved using an eddy viscosity turbulence model (for details see next section).

Influence of Grid Size

The chosen meshing approach for the exhaust hood creates a tetrahedral mesh with inflation layers (prism elements) to accurately resolve the boundary layer. The grid size is chosen to ensure a mesh independent solution. The resulting grid size for the exhaust hood alone is roughly 4M nodes, which is in good agreement with recent publications (e.g. Burton et al. (2013), Stein et al. (2015) etc.) and will therefore not be discussed in more detail here. However, the whole blade row of the last stage is included in the model and the size of the rotor mesh has a large impact on the overall mesh size. As the the flow field within the blading is not of primary interest, but should rather serve to get realistic inlet conditions for the exhaust, this mesh should be as coarse as possible. All secondary flow features will be mixed out at the interface anyway. A node count of 50K nodes per passage is therefore found to be sufficient (roughly 3M nodes for the complete row). This will be shown on a simplified model of the turbine CFD. A single stator vane and rotor blade are coupled to a quasi-2 dimensional (Q2D) slice of the diffuser. The mesh size of the blade passage is varied from 50K nodes to 1M nodes. The mesh resolution of the vane and diffuser are the same as in the 3D CFD. Figure 4 shows the computational domain and a comparison between both mesh sizes for the flow profiles at the diffuser

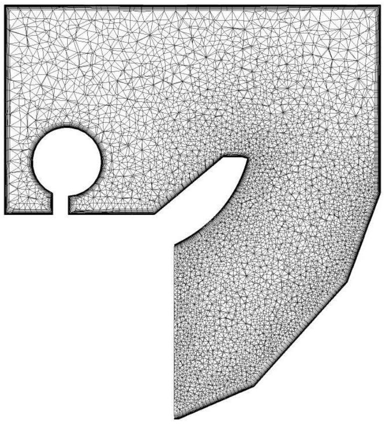

Figure 3: **Mesh resolution**

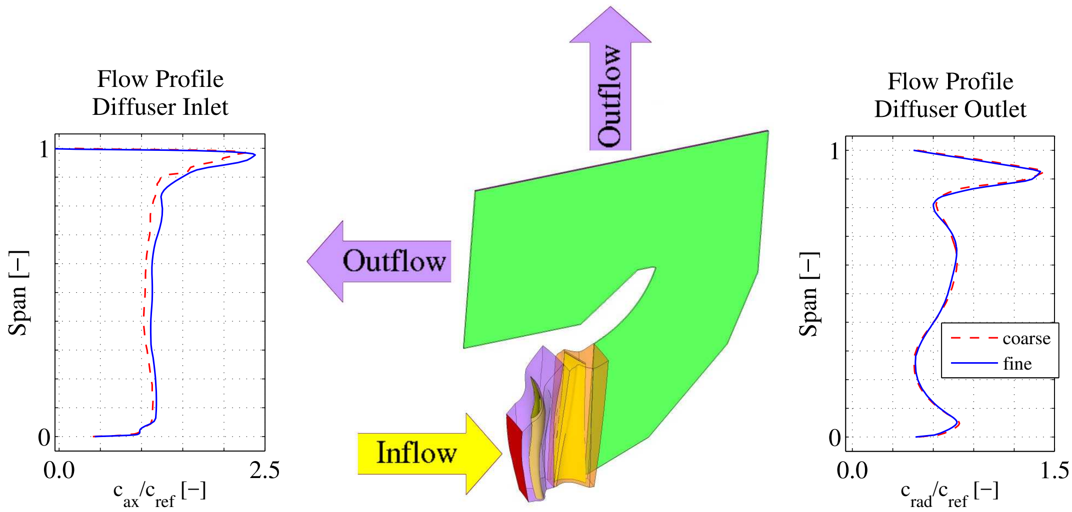

Figure 4: **Influence of blade passage mesh on diffuser flow prediction**

inlet and outlet (i.e. plane E32 and E40 respectively). It can be seen that the mesh resolution of the blade passage is of minor importance for the diffuser flow field. This is in part due to the mixing plane approach used, as flow disturbances in circumferential direction are mixed out. Of course this is also due to the fact that secondary flow effects are of minor importance for blades with large radial extent.

Some Remarks on Turbulence Modeling

First of all, it is well known that the correct prediction of flow separation is a major challenge for all turbulence models. Furthermore, it can be shown that for a diffuser flow without separation, the impact of the turbulence model is only of minor importance. Thus, a diffuser flow far enough away from separation is likely to be correctly predicted by any model. However, as this paper deals with flow close to separation the outcome of the optimisation is of course strongly depending on the applied turbulence model. For the sake of computational simplicity only eddy viscosity turbulence models have been considered in this study and it could indeed be seen that the optimsed diffuser shape depends on the model which is used. Therefore, a turbulence model which predicts an earlier onset of separation will always create a more "conservative" shape. As has been shown by Bardina et al. (1997) the $k - \omega$ based Shear-Stress-Transport (SST) model is favourable in this respect to other eddy viscosity models. This is also the reason why the SST model is applied for all simulations presented here. However, the general approach presented in this paper is independent of the turbulence model used. Therefore, the reader might decide by himself which turbulence model best fits his needs.

Comparison to Measurement

As mentioned above, the flow profiles in E32 and E40 have been measured at different circumferential positions. The exact positions are illustrated in figure 5. Figure 5 also shows the flow profiles downstream of the last stage (i.e. diffuser inflow) as calculated by the CFD

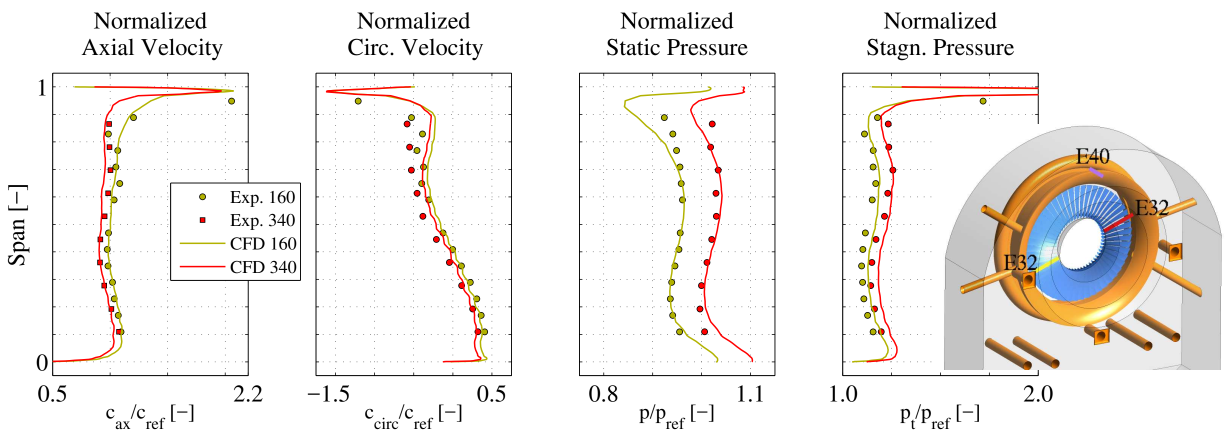

Figure 5: **Comparison of CFD with measurement in plane E32**

compared to the measurements at these two positions. It can be seen that the flow profiles are in very good agreement with the measurements for both circumferential positions. Two conclusions can be drawn from this observation. Firstly, it can be confirmed that the coarse mesh of the blade passage still suffice to capture all necessary flow phenomena. Secondly, it can be deduced that the flow asymmetry and thus the flow losses in the exhaust hood are

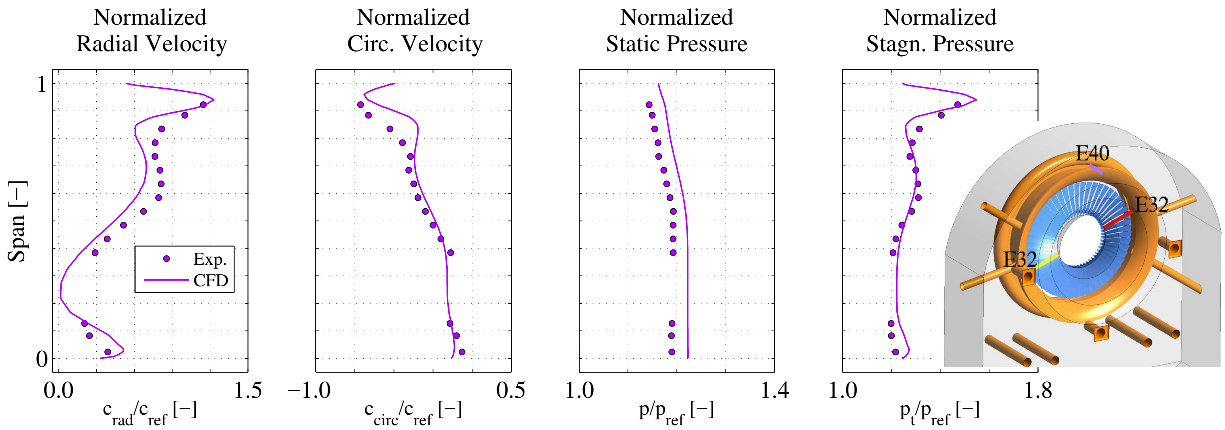

Figure 6: **Comparison of CFD with measurement in plane E40**

correctly predicted. Taking a look at the flow profiles in the diffuser outlet (E40) shown in figure 6 the same conclusion can be drawn. Again, the flow profiles are properly predicted by the CFD. Especially the tip leakage jet shows a good correlation to the measurements. These observations are supported by the wall pressure measurements at the DIS, which are shown in figure 7. Although a slight offset to the measurements can be seen, the overall accuracy is good and the flow asymmetry resulting from the exhaust hood is captured properly. Therefore, it can be concluded that the CFD model is capable to correctly predict the flow asymmetry in the exhaust hood as well as the flow field in the diffuser alone.

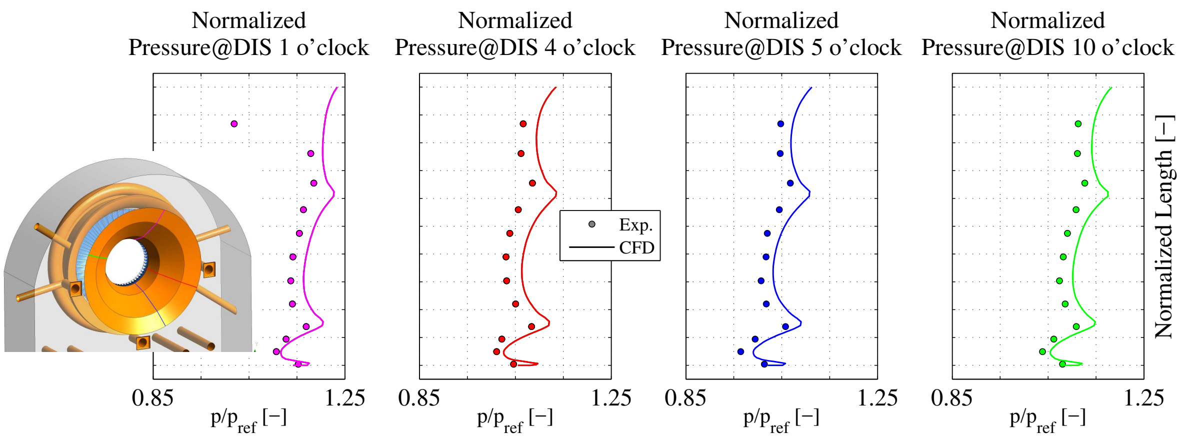

Figure 7: **Comparison of CFD with measurement at diffuser inner shell**

NUMERICAL MODEL PART 2 AND OPTIMISATION APPROACH

The intention of this paper is to create an optimised diffuser shape taking the flow asymmetry due to the exhaust hood into account. The numerical model presented above could in fact be shown to predict the flow accurately in an exhaust hood and would therefore be suitable for this purpose. Though, for an optimisation of the geometry the computational effort of this model is too high to be conducted during the design process of a new turbine or within a retrofit project. Hence, a simplified but still accurate model has to be derived for this purpose. The geometry chosen to verify this approach is the generic exhaust hood geometry published by Burton et al. (2012). Though, a blade design different to the one presented by Burton has been used.

Numerical Model

Basically, the chosen approach to calculate the diffuser flow has already been described by Polklas (2004). The main idea is that the flow field in the diffuser alone can be approximated by a single passage CFD (stator vane and rotor blade) connected to a Q2D diffuser segment. Although the interaction with the exhaust hood is missed this way, the simulation can be adapted to match the flow conditions in the diffuser at least for the 12 o'clock and 6 o'clock position of the exhaust hood. The computational domain for this approach is the same as the one shown in figure 4. It consiststs of a single passage of the stator vane and the rotor blade. The two domains are connected via a mixing plane. The diffuser domain is Q2D and coupled to the rotor domain using a mixing plane. The node count of the stator and rotor domain is comparable to the one used in the whole exhaust CFD. The mesh of the diffuser domain is reasonably fine ($y^+_{\text{max}} < 10$ and at least 15 elements in the boundary layer) to predict any flow separation accurately. As the diffuser is Q2D with periodic boundaries, only one element in circumferential direction has to be modeled. This way it is possible to keep the total node count at roughly 150.000, which is sufficiently small for an optimisation. Boundary conditions are again stagnation pressure and temperature at the inlet and an average static pressure at one of the two outlets. At the other outlet a mass flow rate is prescribed. The mass flow at the outlet boundaries is adjusted to match the pressure field in the diffuser outlet section to the one of the whole exhaust CFD. This way the simplified model can be calibrated to match the real flow field quite well. Also, the same fluid properties as used in the whole exhaust CFD and have been applied. Figure 8 shows a comparison of the flow profiles in the diffuser inlet and outlet for the datum geometry. It can be seen that the Q2D approach is in good agreement with the whole exhaust CFD.

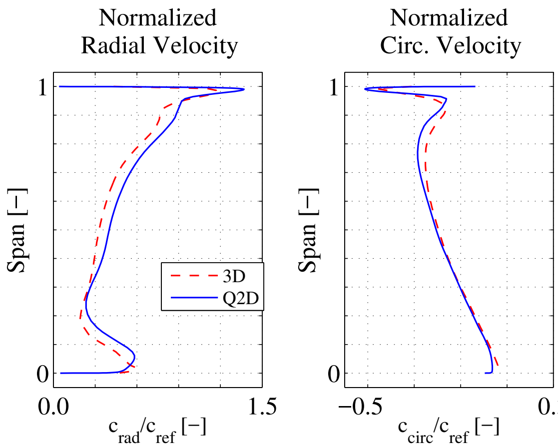

Figure 8: **Comparison of Q2D to 3D approach for 12 o'clock position**

Parametric Model

The parametrisation of the geometry is an important if not the most important contributor to a successful optimisation. It has to ensure that only reasonable geometries are created without restricting the results too much. Also the number of parameters should be kept to an acceptable number. This for example is the reason why higher order polynomials are often a bad choice as they tend to give unusable "wavy" geometries. Lower order polynomials however are often too restricting. Thus, for the present study a spline based approach to represent the diffuser geometry is chosen as shown in figure 9. The geometry consits of two Bézier curves. The starting point and inclination are prescribed by the last stage blade. The inclination

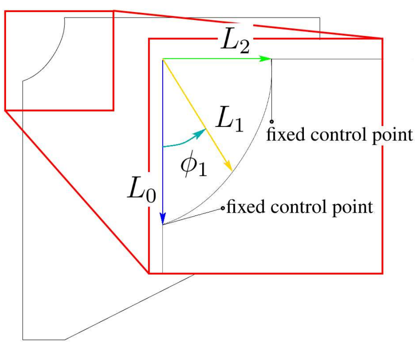

Figure 9: **Parametric model**

at the diffuser outlet is fixed in a purely radial direction. The free parameters which can be changed are therefore the lengths L_0, L_1 and L_2. Additionally, the angle ϕ_1 can be varied. As the investigated exhaust hood is a very "tight" geometry with the hood very close to the diffuser at 12 o'clock, it was decided to only allow a minor increase of the diffuser height at this position. Consequently, the parameter L_0 is almost constant during the optimisation at the 12 o'clock position.

Optimisation Approach

The optimisation for the diffuser sections at the 12 o'clock and the 6 o'clock position will be carried out independently. From these results a final 3D geometry will be created by linear interpolation for all intermediate positions around the circumference. However, the general approach is of course the same for both positions. The first step in the optimisation is to define the problem (i.e. objective function, constraints etc.) and create an intial design. The objective function for this optimisation is the pressure recovery defined by

$$c_p = \frac{p_{\text{out}} - p_{\text{in}}}{p_{t,\text{in}} - p_{\text{in}}} \tag{1}$$

which should be maximized. Apparently, this would not result in a very robust design with regard to flow separation as the highest pressure recovery is expected to be close to separation. Thus, to add a certain degree of robustness to the design the wall shear is used as an additional objective which should be maximized and is furthermore not allowed to drop below a certain threshold. As an initial design the datum diffuser geometry is approximated by the parametric model. The next step is to create a first set of samples by Design of Experiment (DOE) techniques for a given design space. For this investigation, an optimized Latin hypercube sampling, which has been optimized so that the correlation between the inputs is minimized, is used. A first set of samples (e.g. n=50) is now created and computed with the Q2D CFD model. The results are used to train a meta model. The meta model used here is a hybrid model combining a Kriging and a moving least square model. Details of the meta model are discussed in Cremanns et al. (2016). Usually the first set of samples will not result in a meta model having a good enough prognosis quality to carry out an optimisation. This is mainly due to the fact that the diffuser problem is discontinuous as the pressure recovery will decrease abruptly once flow separation occurs. The technique of kriging is known to perform poorly when the data involves localized effects such as discontinuities. However, as only

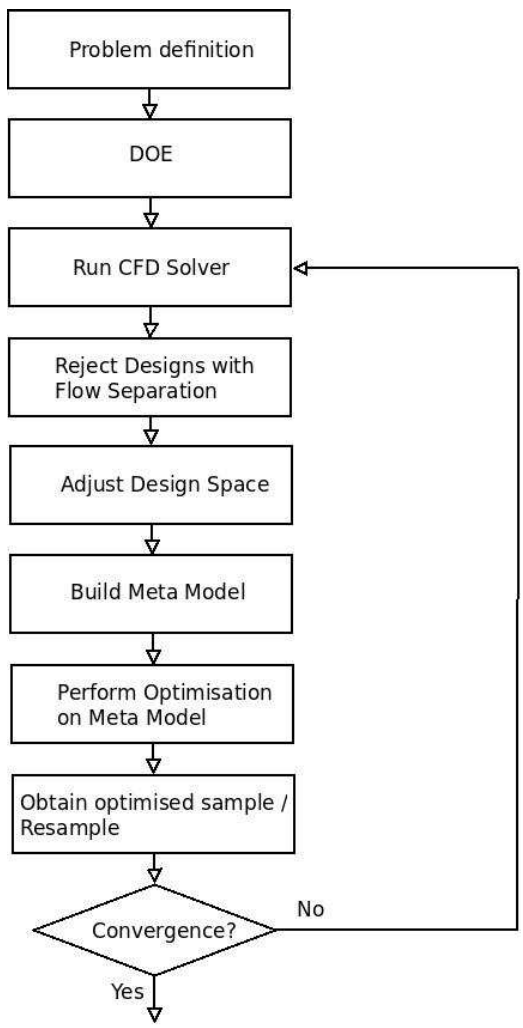

Figure 10: **Optimisation procedure**

designs without flow separation are of interest, all designs which fail to fulfill this requirement can be rejected. This way it is possible to circumvent the discontinuities. Ideally, the failed designs can be used to limit the design space for further optimisation iterations. One has to be aware that depending on the initial design space the majority of the samples must be rejected. With the improved meta model it is now possible to carry out the optimisation. Depending on the result of the optimisation it might be necessary to increase the design space if the calculated optimum is on or close to the limits of the initial design space. For the new design space a second DOE is carried out. This iteration should be continued until no reasonable change for the meta model can be found. Usually an overall amount of 200 samples (including those samples that show flow separation) is sufficient to create a good meta model. The general procedure is shown in figure 10.

OPTIMISATION RESULTS

As mentioned above two contrary objective functions are used in the optimisation, i.e. the pressure recovery and the wall shear, which should both be maximized. Consequently, the result of the optimisation is a pareto front. For the final evaluation two designs for the 6 o'clock and 12

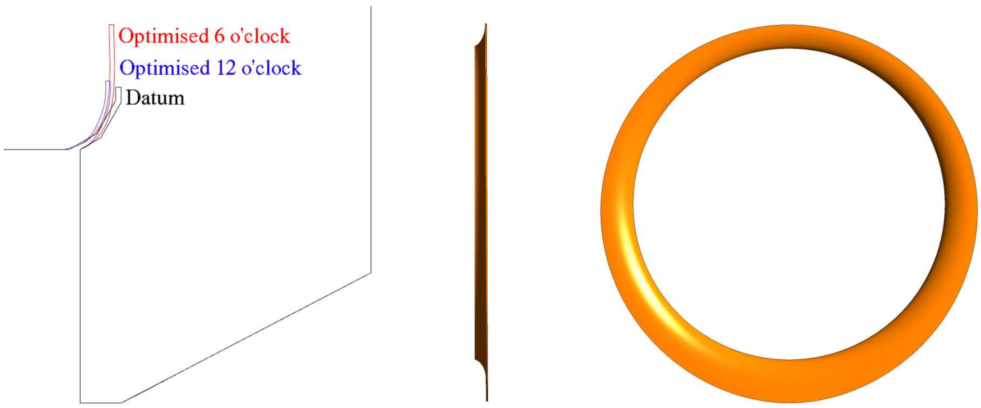

Figure 11: **Geometry of optimised diffuser shape**

o'clock have been chosen with highest c_p-values. The diffuser shape for both positions is shown in figure 11 in comparison to the datum design. Also two views of the 3D shape are shown.

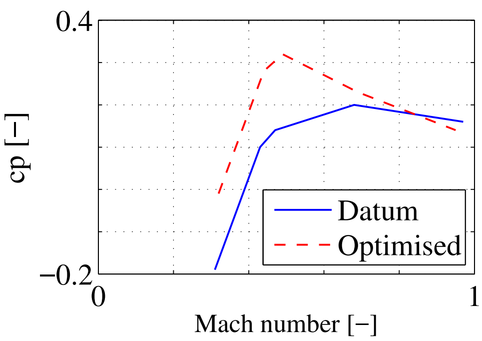

Figure 12: **Pressure recovery**

The design is evaluated not only at the design point but also for off-design operation. Figure 12 shows the pressure recovery versus the Mach number at the diffuser inlet. It can be seen that for the design point a considerable increase in pressure recovery could be reached. This increase in pressure recovery of roughly $\Delta c_p = 0.17$ results in an increased power output of the last stage of 530 kW for a typical steam power plant with a mass flow of approximately 150 $\frac{\mathrm{kg}}{\mathrm{s}}$ passing the last stage and a gross power output of 600 MW. However, it can also be seen that there is an abrupt decrease in the c_p curve. Taking a look at the flow field for two of the simulated operating points it can be seen that for higher flow velocities a large separation is formed. Altough, a separation is also present in the datum design, the separation gets even larger in the optimised design. This again, results in a lower c_p value for the optimised design.

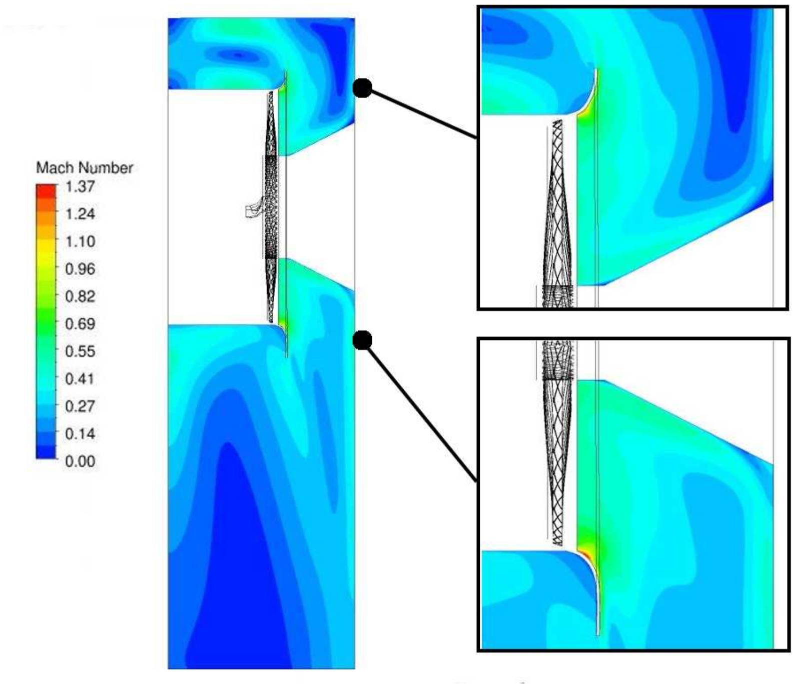

Figure 13: **Flow field of optimised design**

CONCLUSIONS

In the first part of the present study a numerical model is presented which allows for an accurate prediction of the flow within a low pressure steam turbine exhaust hood. The model has been validated with measurements gained from a model turbine. In the second part of the paper a simplified numerical model has been derived from the former model which allows to predict the flow field at two distinct positions in the exhaust hood with a response time short enough to train a meta model in a feasible time. This meta model is then used to carry out an optimisation of the diffuser flow guide. The results of this optimisation are again scrutinized with the help of the validated model from the first part of the paper. It can be shown that a considerable performance increase can be achieved with the optimised design. The cause of the large flow separation in off-design operation is an object for further research to keep the pressure recovery over the whole operational range on a high level.

References

Bardina, J., Huang, P., and Coakley, T. (1997). Turbulence Modeling, Validation, Testing and Development. *NASA Technical Memorandum 110466*.

Burton, Z., Hogg, S., and Ingram, G. (2012). A Generic Low Pressure Exhaust Diffuser for Steam Turbine Research. In *Proceedings of ASME Turbo Expo*. GT2012-68485.

Burton, Z., Ingram, G., and Hogg, S. (2013). A Novel Method of Coupling the Steam Turbine Exhaust Hood and the Last Stage Blades Using the Non-Linear Harmonic Method. In *Proceedings of ASME Turbo Expo*. GT2013-94184.

Cremanns, C., Roos, D., Hecker, S., Dumstorff, P., Almstedt, H., and Musch, C. (2016). Efficient Multi-Objective Optimization of Labyrinth Seal Leakage in Steam Turbines Based on Hybrid Surrogate Models. In *Proceedings of ASME Turbo Expo*. GT2016-57457.

Finzel, C., Schatz, M., Casey, M., and Gloss, D. (2011). Experimental Investigation of Geometrical Parameters on the Pressure Recovery of Low Pressure Steam Turbine Exhaust Hoods. In *Proceedings of ASME Turbo Expo*. GT2011-45302.

Fu, J. and Liu, J. (2010). Investigation of Influential Factors on the Aerodynamic Performance of a Steam Turbine Exhaust System. In *Proceedings of ASME Turbo Expo*. GT2010-22316.

Havakechian, S. and Denton, J. (2015). 3D Blade Stacking Strategies and Understanding of Flow Physics in Low Pressure Steam Turbines: Part I – 3D Stacking Mechanisms. In *Proceedings of ASME Turbo Expo*. GT2015-42591.

Hirschmann, A., Volkmer, S., Schatz, M., Finzel, C., Casey, M., and Montgomery, M. (2010). The Influence of the Total Pressure Profile on the Performance of Axial Gas Turbine Diffusers. In *Proceedings of ASME Turbo Expo*. GT2010-22481.

Li, Z., Li, J., Yan, X., Feng, Z., Ohyama, H., and Zhang, M. (2012). Investigations on the Flow Pattern and Aerodynamic Performance of Last Stage and Exhaust Hood for Large Power Steam Turbines. In *Proceedings of ASME Turbo Expo*. GT2012-69291.

McDonald, A., Fox, R., and Dewonstine, R. (1971). Effects of Swirling Flow on Pressure Recovery in Conical Diffusers. *AIAA Journal*, 9(10):2014–2018.

Musch, C., Stüer, H., and Hermle, G. (2013). Optimization Strategy for a Coupled Design of the Last Stage and Successive Diffuser in a Low Pressure Steam Turbine. *Journal of Turbomachinery*, 135(1):011013–7.

Nicoll, W. and Ramparian, B. (1970). Performance of Conical Diffusers With Annular Injection at Inlet. *Journal of Basic Engineering*, 92:827–835.

Polklas, T. (2004). *Entwicklung eines numerischen Verfahrens zur strömungsmechanischen Auslegung des Abströmgehäuses einer Niederdruck-Dampfturbine*. PhD thesis, Universität Duisburg-Essen.

Stein, P., Pfoster, C., Sell, M., Galpin, P., and Hansen, T. (2015). CFD Modeling of Low Pressure Steam Turbine Radial Diffuser Flow by Using a Novel Multiple Mixing Plane Based Coupling: Simulation and Validation. In *Proceedings of ASME Turbo Expo*. GT2015-42632.

Taylor, D., Singh, G., Hemsley, P., and Claridge, M. (2016). Parametric Experimental and Numerical Study of LP Diffuser and Exhaust Hoods. In *Proceedings of ASME Turbo Expo*. GT2016-56640.

Verstraete, T., Prinsier, J., Sante, A. D., Gatta, S., and Cosi, L. (2012). Design Optimization of a Low Pressure Steam Turbine Radial Diffuser Using an Evolutionary Algorithm and 3D CFD. In *Proceedings of ASME Turbo Expo*. GT2012-69515.

Völker, L. (2006). *Neue Aspekte der aerodynamischen Gestaltung von Niederdruck-Endstufen-Beschaufelungen*. PhD Thesis, Universität Stuttgart.

Wang, H., Zhu, X., and Du, Z. (2010). Aerodynamic Optimization for Low Pressure Turbine Exhaust Hood Using Kriging Surrogate Model. *Int. Commun. Heat Mass Transfer*, 10(8):998–1003.

Yin, M., Yang, C., Meng, L., Yan, W., Zhuhai, Z., Li, J., and Feng, Z. (2016). Numerical Analysis on the Swirl Flows in the Exhaust Hood of Steam Turbine and Optimization Design. In *Proceedings of ASME Turbo Expo*. GT2016-56063.