Turbulence Resolving Flow Simulations of a Francis Turbine in Part Load using Highly Parallel CFD Simulations

Timo Krappel¹, Stefan Riedelbauch¹, Roland Jester-Zuerker², Alexander Jung², Benedikt Flurl¹, Friedeman Unger¹ and Paul Galpin⁴

¹Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart, Pfaffenwaldring 10, 70550 Stuttgart, Germany
²Voith Hydro Holding GmbH & Co. KG, Alexanderstraße 11, 89522 Heidenheim, Germany
³ANSYS Germany GmbH, Staudenfeldweg 20, 83624 Otterfing, Germany
⁴ANSYS Canada Ltd, 283 Northfield Drive E, Waterloo, ON, Canada N2J 4G8

E-mail: timo.krappel@ihs.uni-stuttgart.de

Abstract. The operation of Francis turbines in part load conditions causes high fluctuations and dynamic loads in the turbine and especially in the draft tube. At the hub of the runner outlet a rotating vortex rope within a low pressure zone arises and propagates into the draft tube cone. The investigated part load operating point is at about 72% discharge of best efficiency. To reduce the possible influence of boundary conditions on the solution, a flow simulation of a complete Francis turbine is conducted consisting of spiral case, stay and guide vanes, runner and draft tube. As the flow has a strong swirling component for the chosen operating point, it is very challenging to accurately predict the flow and in particular the flow losses in the diffusor. The goal of this study is to reach significantly better numerical prediction of this flow type. This is achieved by an improved resolution of small turbulent structures. Therefore, the Scale Adaptive Simulation SAS-SST turbulence model – a scale resolving turbulence model – is applied and compared to the widely used RANS-SST turbulence model. The largest mesh contains 300 million elements, which achieves LES-like resolution throughout much of the computational domain. The simulations are evaluated in terms of the hydraulic losses in the machine, evaluation of the velocity field, pressure oscillations in the draft tube and visual comparisons of turbulent flow structures. A pre-release version of ANSYS CFX 17.0 is used in this paper, as this CFD solver has a parallel performance up to several hundreds of millions of mesh elements using thousands of cores for this application which includes a transient rotor-stator interface to support the relative motion between the runner and the stationary portions of the water turbine.

1. Introduction

Due to the liberalization of the power market and due to an increasing amount of renewable energy, like wind and solar energy, hydro power plants have to operate over a wider range. It is important to reach a better understanding of the part load operating point of a low head high specific speed Francis turbine, which is the focus of this paper. At the same time, computational resources have increased, so that it is possible to conduct flow simulations with several hundreds of millions of mesh elements using thousands of cores in parallel [10][18].
The part load operating point ($Q = 0.72 Q_{opt}$ and $H = 0.9 H_{opt}$) is dominated by the so called vortex rope in the draft tube, which evolves due to a typical distribution of the velocity components at the runner outlet. The velocity distribution is characterized by a high swirling and axial component at higher radii. The flow field is transient, highly complex in nature and three-dimensional, making it challenging, if not impossible, to obtain reliably and predictive steady-state simulations of such an operating point [11]. The resolution of more turbulent details is necessary and realized by a scale-resolving simulation (SRS) approach, namely using the scale-adaptive simulation (SAS) turbulence model [3][4][16].

There are several publications showing that the SAS-approach has proved to be beneficial over a typical RANS-approach for the simulation of hydraulic machines: Investigations of a Francis turbine at full load, part load and deep part load [14] and of a Kaplan turbine at different operating points [6] or for the same turbine and same operating point [11].

All these simulations deal with meshes in the order of 10’s of millions of elements analyzing different operating points. In contrast to that, goal of this paper is the usage of larger mesh sizes resolving more details of the flow including small turbulent structures.

2. Numerical modelling

2.1. Flow solver
Simulations were performed using a pre-release version of ANSYS CFX 17.0 [1]. The flow solver is based on an implicit pressure-based formulation with the unknowns located at the element vertices. Collocated control volumes are formed around each vertex from the element mesh-dual, as part of the element flux assembly process. A coupled algebraic multigrid (AMG) linear solver [19] is used with an ILU base smoother. This ensures essentially linear scalability of the linear solver effort as a function of mesh size.

2.2. Turbulence modelling
This paper focuses on unsteady simulations, in order to resolve the flow physics with the highest accuracy possible. In all presented cases either the URANS-SST [15] or the SAS-SST [16] turbulence models were applied. Classical URANS turbulence models cannot provide any spectral content even with a mesh and time step resolution which would otherwise be sufficient for this purpose. This limitation of RANS approaches can be overcome by the use of a scale resolving turbulence model, such as the SAS-SST model.

The SAS-SST model behaves like a standard RANS model for steady flows but can capture broadband turbulence spectrum for unstable flows. As the turbulent eddy viscosity is reduced, a high wave number limiter – based on the LES-WALE model [17] – is applied. If the high wave number limiter is reached, the turbulence model behaves similar to the LES model.

2.3. Discretization schemes
For temporal discretization, an implicit second order backward Euler scheme was used for all transport equations. For spatial discretization, the error is dominated by the convection scheme error. In this work the choice of the discretization scheme used for the convection term is shown to have a significant impact on the prediction accuracy and flow character, due to the sensitivity of the scale resolving turbulence scheme to the level of numerical dissipation. Results from simulations with the hybrid convection scheme (hybCon) [21], a combination of the high-resolution scheme (HR) [2] and the central differencing scheme (CD), and with the bounded central difference scheme (BCD) [13] are shown and compared. Both of these convection schemes are formally second order accurate, but use different numerical strategies to control any tendency for unphysical numerical oscillations in the solution.
2.4. Parallelization Strategy

The CFD solver is highly optimized for large scale parallel systems using the SPMD (Single Program Multiple Data) parallelization approach, combined with common domain decomposition methods (METIS [7], others). The pre- and post-processing of the simulations, as well as the physical models, the linear solution method, and the file I/O are all not affected by the parallel run mode of the solver. The impact of the parallel run mode on the simulation setup has been minimized.

All parts of the solver have been carefully parallelized to avoid serial bottlenecks as much as possible. The domain decomposition topology is achieved with an upfront partitioning run. A special solver binary using 64bit integer values is available which has been used successfully for the domain decomposition of cases with more than 1 billion mesh vertices.

The SPMD parallel model has been strictly followed in the CFD solver, resulting in a nearly perfect balance of memory between the N parallel processes. Although the master process is responsible for a small number of unique tasks (file IO, diagnostics, simulation control), there is no requirement for a “fat head node” which makes the solver ideal on massively parallel systems with symmetrically distributed memory (e.g. cluster systems).

Over the last years, extensive improvements have been implemented based on the experience with the large and complex turbine simulation as described in this present paper. This includes the efficient usage of MPI collective routines, IO improvements, new communication methods as well as hierarchical AMG collection strategies in the linear solver. In addition, there is a special parallel complication for transient simulations with moving mesh interfaces (Transient Rotor Stator model), related to the large overlapping mesh regions between neighboring partitions resulting from the arbitrary relative position between stationary and rotating domains. Reducing the negative impact of those overlap regions on the parallel performance in the equation assembly, as well as the linear solution, is important to maintain parallel performance. Good scaling of complex simulations with multiple frames of reference can now be achieved for thousands of partitions. In addition, the solver supports massively parallel Cray computers using the native Cray MPI for interprocess communication. These recent improvements in scalability in fact enabled this project otherwise the wall clock run times were untenable.

2.5. Details of simulation configuration and code performance

The simulation domain consists of the spiral casing, stay and guide vanes, the runner and the draft tube with an expansion tank (see Figure 1). The smallest computational mesh has 16 million mesh nodes in total (see Table 1) with a near-wall resolution in the range of $y^+ = 15-30$. All of the larger meshes have a near-wall resolution of $y^+ = 1$. The mesh refinement between the 50 million and 300 million nodes strongly focuses on the draft tube domain, almost reaching LES-like resolution in the boundary layer. The upstream domains are somewhat less refined by choice, as the most complex flow is expected in the draft tube region. The time-step size is chosen to keep the Courant number roughly below one in the whole computational domain.

Figure 1. Picture of the whole computational domain with surface-mesh (left) and a zoom-in showing the evaluation lines in the draft tube region of the domain and monitor points D and G (right)
Table 1. Description of meshes with node numbers per component (M=million) and time step size

| Name | Spiral casing | Stay and guide vanes | Runner | Draft tube | Time steps/revolution |
|------|---------------|----------------------|--------|------------|-----------------------|
| 16M  | 1.0M          | 3.7M                 | 3.7M   | 8.1M       | 180                   |
| 50M  | 4.5M          | 10.2M                | 13.5M  | 22.1M      | 720                   |
| 150M | 7.3M          | 18.0M                | 29.9M  | 98.8M      | 840                   |
| 300M | 11.8M         | 27.9M                | 55.0M  | 211.6M     | 1000                  |

The strong-scaling speedup test of different versions of ANSYS CFX using 576 to 4032 cores in parallel on the 300M-mesh can be seen in Figure 2. The parallel performance improvement between the different versions is visible. The performance between the versions up to 1000 cores is almost the same, but version 17.0 is seen to maintain parallel scaling up to the highest core count tested. Nevertheless, such a simulation still takes several weeks of wall clock time, due to the large number of time steps per revolution (1000) and the desire to perform up to 50 full revolutions of simulation time. The simulations have been performed at the high performance computing center in Stuttgart (HLRS) on a Cray XC40 cluster.

Figure 2. Wall clock speedup, normalized by the baseline time required for a 576 core simulation, is plotted versus total core count, for different versions of the CFD solver, all for the 300M-mesh simulation performed on a Cray XC40.

3. Results

3.1. Global machine data

The comparison of hydraulic losses of a Francis turbine is a fundamental basis upon which to evaluate the different simulation approaches, including sensitivity of predictions to change in mesh density, discretization schemes and turbulence models. The results can be seen in Figure 3 and are subsequently discussed.

The hydraulic losses of the total machine – between the spiral casing inlet and the draft tube outlet – are clearly higher predicted by the simulations with SST turbulence model. In contrast to that, all simulations with the SAS-turbulence model lead to lower hydraulic losses. The losses obtained with the BCD-scheme are lower than those of the hybrid convection scheme. For both convection schemes using the SAS-turbulence model no strict grid convergence is reached, even for the 300M-mesh, with the lowest values. The trend is: the finer the mesh the lower the losses.

The evaluation of the losses of each component reveals some differences in the composition of the total losses. The results with the SST-model, especially with the coarse mesh, predict the highest losses in all components. This is explained by its dissipative character of a RANS model and its inability to resolve the turbulent flow structures. The result of the 16M-SAS-hybCon-simulation shows higher draft tube losses, which might be caused by the deviant tangential velocity component in the draft tube (see Figure 4). The losses obtained by using the SAS-model decrease with larger
meshes, whereas the hybrid convection scheme seems to predict slightly lower losses than the BCD-scheme.

![Figure 3](image)

**Figure 3.** Hydraulic losses for different simulation approaches: total machine (top, left), SVWG- and RU-component (top, right), DT-component (bottom, left) and Euler head of the runner (bottom, right)

The losses predicted by the BCD-scheme rarely vary for the components of stay- and guide-vanes and runner. In contrast to that, the losses for these components using the hybrid convection scheme decrease for larger meshes. This might be explained by the nature of this model, which combines the high-resolution scheme in the inflow and from the stay and guide vanes on – when turbulent structures are resolved, especially with larger meshes – a pure central differencing scheme (CD) except within the boundary layer. Therefore, the results of the coarser meshes are closer to the results of the SST turbulence model and the results of the finer meshes are closer to the results of the BCD-scheme.

For all meshes there is still a visible offset between the convection schemes for the runner losses. This might be explained by the swirling-component or Euler head \(H_{Eul,i} = u_i c_{i,u}/g\) for position \(i\) at the runner inlet, which is higher for the more dissipative scheme and somewhat decreases for larger mesh sizes. The Euler head at the runner outlet for the 16M-mesh simulations is around 1\% higher than for the simulations on the finer meshes. All these simulations have an almost constant Euler head at the runner outlet, whereby the inflow into the draft tube should be quite similar.

### 3.2. Flow analysis

The velocity on the specified evaluation lines in the cone and diffuser are depicted in Figure 4 and Figure 5 (the positions of the evaluation lines are according Figure 1).

The axial velocity component in the cone shows similar results for all simulation types on the 16M-mesh. The results of the all simulations on the finer meshes are also quite similar but with higher values in the center of the cone, except for the 50M-SST-simulation, which is in between. The tangential velocity component of both SST-simulations and the 16M-BCD-simulation is almost
equally predicted with lower values in the center of the cone. The 16M-hybCon-simulation shows a different behavior with even less swirl in the center. The reason for this might be due to effects in the turbulence model, as later discussed in Section 3.4. The other simulation types also show a quite similar behavior with a higher swirling component in the center of the cone.

![Figure 4](image.png)

**Figure 4.** Velocity distributions in the cone; left: axial velocity component, right: tangential velocity component; negative radius indicates head-water side and positive tail-water side

At the end of the draft tube diffuser, the flow field is separated at the upper wall \( (L/L_{ref} = 1) \) for the SST-simulations and especially for the coarse mesh simulation. The meridional velocity profile of the 16M-SAS-BCD-simulation is less steep and predicts a much higher velocity component at the upper wall. For larger mesh sizes this trend is reduced and the meridional velocity component is closer to zero.

![Figure 5](image.png)

**Figure 5.** Velocity distributions in the diffuser of the stream-wise velocity component for different stream-wise positions

3.3. Vortex rope induced pressure pulsations

The vortex rope induced pressure pulsations resulting from the different simulation approaches are shown in Figure 6. In the upper part of the draft tube cone, the wall-pressure signal has an almost sinusoidal shape, mainly dominated by the first and second mode of the vortex rope.

These frequencies are very similarly predicted by all simulation approaches. The higher frequencies, like the runner blade wakes, depend on the capability to resolve small flow structures. Therefore, only the larger meshes with SAS-model resolve these higher frequencies, namely at \( f / f_r = 13 \) and \( f / f_r = 26 \). For the RANS-simulation these fine flow structures are already dissipated in the flow field at this position.

Experimental investigations were done at the closed loop test rig at the laboratory at the Institute of Fluid Mechanics and Hydraulic Machinery, University of Stuttgart. The wall-pressure pulsation in the draft tube cone in point D is measured with piezo-resistive pressure transducers.
The simulations fit quite well with the experimental results. The shape of the measured low frequency pressure oscillation of the first modes and of the runner blades are quite similar to the shape predicted by the simulations, except for the 50M-SST-simulation. This can also be seen in the FFT-analysis, where the frequency and amplitudes of the first two harmonics match quite well between measurement and numerical results. In the range of $f/f_n = 1.5$ and for the runner blade frequencies, the measurements show somewhat higher amplitudes.

At the end of the draft tube cone the wall-pressure signal is more complex, as it consists of several dominating modes, for example for the first six to nine modes. With finer meshes the upper slope of the signal is not unique and is changing its shape for the different main periods of the vortex rope. The
simulations with the 300M-mesh and SAS-model identify higher frequencies above $f/f_n = 5$. At this position, the vortex rope is well developed and rotates around its own axis, which might lead to these typical frequencies [8][9].

3.4. Evaluation of turbulence and turbulent flow structures

The RANS-SST-simulations show considerably higher values of turbulent eddy viscosity, as expected (see Figure 7). The result of 16M-SAS-hybCon has also quite high values in the cone. This occurs because the hybrid convection scheme is the more dissipative convection scheme. This prevents the SAS-model to switch into the SRS-mode upstream of the cone, but occurs where the vortex rope appears. This might also be the region where the SAS model is part way between RANS mode and SRS-mode [20] – right where the switch into SRS-mode – arises. This phenomenon is undesired in the vortex rope region, which might explain the velocity distribution in the cone. For the larger meshes the dependence between mesh density and eddy viscosity is visible: the higher the mesh density the lower the eddy viscosity. The hybrid convection scheme using the CD-scheme in the draft tube is less dissipative than the BCD-scheme, leading to a further reduction of the eddy viscosity, which is slightly suppressed by the BCD-scheme for the SAS-model. The eddy viscosity ratio for the 300M-SAS-hybCon-simulation is in the range of 10 and 20, indicating more or less LES-like resolution.

![Figure 7. Turbulent eddy viscosity ratio in the draft tube cone (left) and diffuser (right) (legend is the same as in Figure 4)](image)

The predicted flow structures of some selected results are visualized with the Q-criterion [5], as can be seen in Figure 8. The large shape of the vortex rope occurs at the runner outlet and extends into the draft tube cone. The vortex rope decays in the draft tube elbow into smaller turbulent structures propagating into the diffuser. The RANS-approach is merely predicting large flow structures, as expected by its nature. The result of the 16M-SAS-simulation with hybrid convection scheme shows that in the cone only large flow structures can be resolved, as the turbulence model does not switch into SRS-mode. At first in the elbow the smaller turbulent structures are resolved and the eddy viscosity is reduced. In contrast to that, the result of the BCD-scheme already shows in the cone somewhat smaller structures. With larger mesh sizes and a better resolution of the boundary layer, already in the draft tube cone small structures are resolved, for example the runner blade wakes. With the finest 300M-mesh, very fine flow structures are resolved in the entire draft tube. For this mesh size, combined with the hybrid convection scheme, even smaller turbulent flow structures are resolved as with the BCD-scheme.

4. Summary

In this work, a turbulent scale-resolving flow simulation of an entire low head Francis turbine, from upstream of the spiral casing to downstream of the draft tube, at part load operating condition is discussed and presented. As the largest mesh contains more than 300 million elements, a highly parallel CFD-solver was required.
It could be shown that hydraulic head losses show significant mesh dependency, whereas finer meshes lead to reduced loss prediction. The convection scheme influences the Euler head upstream from the runner. The flow field in the draft tube cone mostly depends on the mesh size and less on the numerical scheme. At the end of the draft tube diffuser area, the RANS-simulations predict a clear flow separation, whereas the other numerical approaches do not show a distinct tendency.

The pressure pulsations in the draft tube cone are better predicted by the finer meshes. Especially the finest mesh (300M) reveals a higher frequency content. It could be shown that the pressure pulsations do not only have a sinusoidal shape, but consist of nearly ten dominating modes at the draft tube cone end, where the vortex rope is well shaped.

The evaluation of the turbulent eddy viscosity showed that the RANS-model has large values indicating that this model is dissipative, as expected. In contrast, the SAS-approach can resolve the large eddy flow structures and hence leads to a reduction of the eddy viscosity down to typical LES-like values for the largest mesh. The reduction of the eddy viscosity somewhat depends on how dissipative the formal second order convection schemes are.

In this work, we could show that is necessary to use mesh sizes up to 300 million elements in order to resolve more details of the flow field in the entire hydraulic machine. This can only be achieved by applying transient simulations with scale-resolving turbulence model. As the wall clock time for such a simulation is still very high, also meshes with about 50 million elements deliver acceptable results, except of some details.

Further extensive post-processing analysis will follow to better understand the differences in the results by details of the flow prediction. In order to better classify the results which still might depend on the mesh size, a comparison with measurements will be done. Especially LDA-measurements in the draft tube cone, elbow and diffuser shall allow a detailed comparison with the flow field of the numerical results.

Acknowledgement
The authors would like to thank HLRS Stuttgart for providing computational resources within the Bundesprojekt framework.

References
[1] ANSYS Inc. 2016 ANSYS CFX Version 17.0
[2] Barth TJ and Jesperson DC 1989 The Design and Application of Upwind Schemes on Unstructured Meshes AIAA Paper 89-0366
[3] Egorov Y and Menter F R 2008 Development and Application of SST-SAS Turbulence Model in the DESIDER Project Advances in Hybrid RANS-LES Modelling, Notes on Numerical Fluid Mechanics and Multidisciplinary Design Volume 97 pp 261-270

[4] Egorov Y, Menter F R and Cokljat D 2010 The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions. Part 2: Application to Aerodynamic Flows Journal Flow Turbulence and Combustion 85(1) pp 139-165

[5] Jeong J and Hussain F 1995 On the identification of a vortex Journal of Fluid Mechanics Volume 285 pp 69-94

[6] Jošt D, Škerlavaj A and Lipej A 2012 Numerical flow simulation and efficiency prediction for axial turbines by advanced turbulence models 26th IAHR Symposium on Hydraulic Machinery and Systems Beijing China

[7] Karypis G and Kumar V 1995 MeTiS: Unstructured Graph Partitioning and Sparse Matrix Ordering System University of Minnesota

[8] Kirschner O, Ruprecht A and Göde E 2009 Experimental Investigation of Pressure Pulsation in a simplified Draft Tube Proceedings of the 3rd IAHR International Meeting of the Working Group on Cavitation and Dynamic Problems in Hydraulic Machinery Brno Czech Republic

[9] Koutník J, Krüger K, Pochyly F, Rudolf P and Haban V 2006 On Cavitating Vortex Rope from Stability During Francis Turbine Part Load Operation IAHR International Meeting of Working Group on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems Barcelona Spain June 28-30

[10] Krappel T, Ruprecht A and Riedelbauch S 2015 Turbulence Resolving Flow Simulations of a Francis Turbine with a Commercial CFD Code High Performance Computing in Science and Engineering ‘15 Springer Berlin

[11] Krappel T, Ruprecht A, Riedelbauch S, Jester-Zuerker R and Jung A 2014 Investigation of Francis Turbine Part Load Instabilities using Flow Simulations with a Hybrid RANS-LES Turbulence Model 27th IAHR Symposium on Hydraulic Machinery and Systems Montreal Canada

[12] Leonard BP 1991 The ULTIMATE conservative difference scheme applied to unsteady one-dimensional advection Comp. Methods Appl. Mech. Eng., 88:17–74

[13] Jašak H, Weller H G and Gosman A D 1999 High resolution NVD differencing scheme for arbitrarily unstructured meshes Int. J. Numer. Meth. Fluids, pp. 413 – 449

[14] Magnoli M V and Schilling R 2014 Numerical Simulation of Pressure Pulsations in Francis Turbines with Hybrid Turbulence Models Proceedings of the 10th International ERCOFTAC Symposium on Engineering Turbulence Modelling and Measurements Marbella Spain

[15] Menter F R 1994 Two-equation eddy-viscosity turbulence models for engineering applications AIAA-Journal, 32(8), pp. 1598 - 1605

[16] Menter F R and Egorov Y 2010 The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions, Part 1: Theory and Model Description Journal Flow Turbulence and Combustion 85(1) pp 113-138

[17] Nicoud F and Ducrós F 1999 Subgrid-Scale Stress Modelling Based on the Square of the Velocity Gradient Tensor Flow, Turbulence and Combustion, 62, pp. 183-200

[18] Pacot O, Kato C and Avellan F 2014 High-resolution LES of the rotating stall in a reduced scale model pump-turbine 27th IAHR Symposium on Hydraulic Machinery and Systems Montreal Canada

[19] Raw M J 1996 Robustness of Coupled Algebraic Multigrid for the Navier-Stokes Equations AIAA 96-0297, 34th Aerospace and Sciences Meeting & Exhibit, Reno, USA

[20] Schwamborn D and Strelets M 2012 ATAAC – an EU-project dedicated to hybrid RANS/LES methods Notes on Numerical Fluid Mechanics and Multidisciplinary Design Volume 117 pp 59-75

[21] Strelets M 2001 Detached Eddy Simulation of Massively Separated Flows AIAA Paper 2001-0879, 39th Aerospace Sciences Meeting and Exhibit, Reno, NV