Investigation of different simulation approaches on a high-head Francis turbine and comparison with model test data: Francis-99

Peter Mössinger, Roland Jester-Zürker, Alexander Jung
Voith Hydro Holding GmbH & Co KG, St Pöltener Straße 43, 89522 Heidenheim, Germany
E-mail: peter.moessinger@voith.com

Abstract. Numerical investigations of hydraulic turbo machines under steady-state conditions are state of the art in current product development processes. Nevertheless allow increasing computational resources refined discretization methods, more sophisticated turbulence models and therefore better predictions of results as well as the quantification of existing uncertainties. Single stage investigations are done using in-house tools for meshing and set-up procedure. Beside different model domains and a mesh study to reduce mesh dependencies, the variation of several eddy viscosity and Reynolds stress turbulence models are investigated. All obtained results are compared with available model test data. In addition to global values, measured magnitudes in the vaneless space, at runner blade and draft tube positions in term of pressure and velocity are considered. From there it is possible to estimate the influence and relevance of various model domains depending on different operating points and numerical variations. Good agreement can be found for pressure and velocity measurements with all model configurations and, except the BSL-RSM model, all turbulence models. At part load, deviations in hydraulic efficiency are at a large magnitude, whereas at best efficiency and high load operating point efficiencies are close to the measurement. A consideration of the runner side gap geometry as well as a refined mesh is able to improve the results either in relation to hydraulic efficiency or velocity distribution with the drawbacks of less stable numerics and increasing computational time.

1. Introduction
The increasing availability of computer resources allow the use of more accurate numerical schemes such as advanced turbulence models or higher discretization of the model domain. Therefore the Norwegian University of Science and Technology and Lulea University of Technology jointly are organizing a set of workshops to determine the state of the art of high head Francis turbine simulation. As the first of three upcoming workshops the present one is aimed at steady state simulations of a model turbine in three different operating points. In the pending workshops more sophisticated investigations like transient operating conditions and fluid structure interactions are investigated.

Voith Hydro, as one of the world’s leading manufactures in the field of hydropower equipment and services, gathered a broad experience in hydraulic model test and numerical simulations. Increasing demands of computational fluid dynamics (CFD) to support the hydraulic development process lead to a continuous improvement of computer resources and simulation methods. To validate the current development process the provided geometry is used as an input for the in-house mesh and setup procedure. The numerical results are validated with experimental data and can also be compared with the results of other participants.
2. Experimental Setup
2.1. model machine
The model machine used for this test case is the Tokke model, designed at the Norwegian University of Science and Technology (NTNU) and whose prototype turbine \((H = 377 \, m, P = 110 \, MW)\) is operating at the Tokke power plant in Telemark, Norway. The turbine, operating at a test rig with open loop water circuit, is scaled down to 1 : 5.1 model size [3]. Beside the runner with a total of 15 splitters and 15 full length blades the model includes 14 stay vanes inside the volute casing, 28 guide vanes and an elbow type draft tube. Figure 1 shows the meridional section of the model. The sketch also indicates the positions for pressure and velocity measurement.

Figure 1: model machine, location of the pressure sensors and LDA measurement sections [3]

2.2. measurement
Three operating conditions were tested, which are visualized in the hill chart (cf. figure 2) and summarized in table 1. Beside the operation point for best efficiency (BEP), a point with very low discharge at part load (PL) and a high load point (HL) was chosen.

Figure 2: hill chart of the high head Francis turbine model [3]
Table 1: operating points and experimental data for pressure and velocity (*) measurement

| Parameter           | symbol | unit | PL  | BEP  | BEP* | HL  | HL* |
|---------------------|--------|------|-----|------|------|-----|-----|
| Net head            | H      | m    | 12.29 | 11.91 | 12.77 | 11.84 | 12.61 |
| Flow rate           | Q      | m³/s | 0.071 | 0.203 | 0.21  | 0.221 | 0.23  |
| Runner angular speed| n      | 1/s  | 6.77 | 5.59 | 5.74  | 6.16  | 6.34  |
| Guide vane angle    | γ      | °    | 3.91 | 9.84 | 12.44 |
| Efficiency          | ηₕ    | %    | 71.7/72.5* | 92.6 | 92.4  | 90.7 | 91.0 |
| Speed factor        | nₑd   | -    | 0.22 | 0.18 | 0.20  |
| Discharge factor    | Qₑd   | -    | 0.05 | 0.15 | 0.17  |

Time dependent pressure measurement was realized in both the rotating and stationary frame at six positions according to figure 1. Additional velocity measurements of axial and tangential velocity components were carried out, using a laser Doppler anemometer (LDA) along two horizontal lines in the draft tube (cf. figure 1). Due to induced vibrations in the test rig at BEP and HL which caused interferences with the measurements it was necessary to change the absolute values of head, discharge and runner speed [3]. To maintain the required operating point the non-dimensional quantities nₑd and Qₑd were kept constant (cf. table 1).

3. Numerical Setting

3.1. solution setup

All simulations are performed with the software ANSYS CFX v15.0. CFX is an established and well validated fluid solver and allows very high accuracy in the prediction of complex turbulent flows [2]. It uses a finite volume based discretization scheme up to second order accuracy for convective fluxes and truly second order accuracy for diffusive fluxes. Due to the demanding geometry, particular the guide vane fillets and the runner splitter blades, which results in a reduced grid quality, a physical timescale control of Δt = 10⁻³ s is necessary to gain numerical stability. Convergence is achieved with root mean square residuals of pressure, mass-momentum and turbulence parameters of r ≤ 10⁻³ and a total of iteration steps with elapsed pseudo time which exceeds the total domain advection time of factor 5 for simulations with spiral case and factor 10 for all other simulation runs.

During the analyses different kinds of two-equation eddy viscosity (k−ε and k−ω−SST [5, 8, 13]) and Reynolds stress models (BSL-EARSM, BSL-RSM [1, 11]) are applied. As a main difference between these two approaches, the two-equation models using an eddy viscosity hypothesis for the Reynolds stresses, which relates them linearly to the mean velocity gradient:

\[ u_i u_j = \nu_T S_{ij} - \frac{2}{3} \delta_{ij} \]  

while the explicit Reynolds stress model (EARSM) gives a nonlinear relation between the Reynolds stresses and the mean strain-rate, whereas the Reynolds stress model (RSM) solves all additional six terms.

3.2. mesh and boundary conditions

In steady-state simulations the interface (IF) between the stationary and the rotating frame can be modeled either using a circumferential averaging of the flow variables (stage interface) or by changing the reference frame while the relative orientation of the components across the interface is fixed (frozen rotor). The following computations are done using a stage interface frame.
change model between the stationary frame (guide vanes) and the runner. By circumferentially averaging the flow variables it is sufficient to model only one passage of stay vane, wicket gate and runner, applying rotational periodic interfaces. For the present case narrow gaps between stay and guide vanes impede a combined meshing of the two stationary blade rows in one domain. Meshing is done with support of an in-house tool, using scaled geometry and block-structured hexahedral cells, ensuring constant mesh conditions when varying the guide vane angle and furthermore allowing a rapid change of mesh parameters or interface positions. In this case an additional frozen rotor interface is applied between the two domains (cf. figure 3a). Figure 3b shows the respective mesh. While the runner is modeled in the same passage-periodic way, a full draft tube mesh is required (cf. figure 4b,c). The stay vanes in the present geometry extend far in the spiral case which prevents a separate modeling. Therefore the spiral case together with the stay vanes are meshed manually and, to avoid the frozen rotor interface, coupled with all 28 guide vanes by a general grid interface (GGI) for the two non-matching grids (cf. figure 4a). Hence four different domains lead to a variation of possible geometry combinations, summarized in table 2. As a basic geometry, $G_1$ contains all domains except for the spiral case. This configuration is expected to yield the best results for a numerical simulation. The influence of the spiral case is investigated with variant $G_2$. The draft tube plays a major role in terms of hydraulic efficiency and velocity distribution beneath the runner. Variant $G_3$ is used to investigate this influence. A transition from the runner passage to the draft tube is realized by circumferentially averaging the flow variables. Therefore in variant $G_4$ a shifted interface
beneath both measurement planes allows the evaluation of a direct runner outflow. The meshed geometries result in grid densities which are summarized in table 3. Beside the basic coarse grid \(M_2\), with a total of 3.6 Mio. nodes the necessity of a combined spiral case, stay vane mesh results in a mesh \(M_1\) with 20 Mio. nodes. To evaluate the grid sensitivity a refined mesh \(M_3\) is used after the analysis of all basic calculations and having identified the best configuration. To allow the use of in-house pre- and post-processing procedures the runner rotational speed is kept constant at \(n_{\text{BEP}} = 5,59\ \text{s}^{-1}\). Flow rate and head are adjusted accordingly in order to achieve constant non-dimensional values of \(n_{ed}\) and \(Q_{ed}\).

\[
n_{ed} = \frac{nD}{\sqrt{gH}} \quad \text{(2)}
\]

\[
Q_{ed} = \frac{Q}{D^2 \sqrt{gH}} \quad \text{(3)}
\]

To adjust the particular operating point the measured head is multiplied with the efficiency, yielding to the so-called Euler head:

\[
H_{\text{eul,exp}} = H \cdot \eta_{h,\text{exp}} \quad \text{(4)}
\]

where the Euler head from the simulation is calculated with the flow velocities at the runner entrance and exit:

\[
H_{\text{eul,CFD}} = \frac{1}{g} (u_2 c_{2u} - u_1 c_{1u}) \quad \text{(5)}
\]

Table 3: used grid densities and resulting quality parameters

| Grid type               | \(M_1\)         | \(M_2\)       | \(M_3\)       |
|------------------------|-----------------|---------------|---------------|
| Distributor+spiral case| 17,969,250      | -             | -             |
| Distributor            | -               | 1,343,873     | 3,700,014     |
| Runner                 | 636,889         | 636,889       | 4,124,348     |
| Draft tube             | 1,633,239       | 1,633,239     | 3,150,259     |
| Total nodes (\(\times 10^6\)) | 20.24 | 3.61 | 10.97 |
| Min. element angle     | 8.3\(^{\circ}\) | 15.2\(^{\circ}\) | 13.3\(^{\circ}\) |
| Max. aspect ratio      | 290             | 254           | 565           |
| Max. expansion factor  | 789             | 101           | 166           |
| Mean \(y^+\)           | 53              | 51            | 32            |
By correcting the guide vane angle in a manner to match the required Euler head within ±1% the similarity between the velocity fields before and after the runner for the experiment and simulation is ensured. As boundary condition a mass flow at the inflow according to table 1 and corrected with eq. 3 is applied. Additionally the inlet flow angle is set to 8.5° and turbulence parameters are likewise set to the fixed values of 5% intensity and an eddy length scale of 10⁻² m. The domain outlet is defined with a zero static pressure condition allowing backflow by switching to total pressure based on the normal component of velocity.

4. Results

4.1. average pressure

Six pressure taps along the hydraulic waterway (cf. figure 1) measured the pressure at all three operating points. Due to the arbitrary pressure level of the numerical calculation the first sensor location at the runner blade pressure side P42 is used to identify the absolute pressure level. Figure 5 summarizes the percentage differences between the measured and the calculated pressure. The simulations are based on the basic domain (G₁) with the coarser grid (M₂) and the k-ω-SST turbulence model. In general a good agreement between the experimental

![Figure 5: comparison between numerical and experimental pressure at five positions](image)

and numerical results is found. For the blade pressure taps the differences amount to ±5%, whereas in the draft tube at all operating points the pressure is calculated about −7% lower than measured. In the vaneless space deviations up to −15% are found. A local mesh improvement has no considerable influence. The procedure of an adjusted wicket gate angle to reach the required Euler head leads to a slightly different pressure distribution in this area and thus may contribute to the deviations. It is expected that pressure levels are not varying in a significant manner for different model domains. The same applies by altering turbulence models, as already concluded by Trivedi et al. [10]. Furthermore, according to table 1, operating points for pressure measurements differ from those of the velocity measurements. Therefore the focus on proceeding numerical simulations lies in the operating conditions of the velocity measurements and comparing the resulting velocity distribution along the draft tube sections.

4.2. velocity distribution

Velocity measurements are carried out along two horizontal sections in the draft tube according to figure 1. The components available are in axial \(c_{ax}\) and tangential \(c_u\) direction. For a comparison with numerical data, velocities are normalized with the discharge at the respective operating point and the draft tube inlet area.

\[
c_0 = \frac{Q_{in,DT}}{A_{in,DT}}
\]
A first evaluation compares the different turbulence models. As in the previous chapter, simulations were done for the basic domain on the coarser grid. Further investigations evaluate the influence of different domains and a second, refined mesh. For part load operating conditions (cf. figure 6) it is found a satisfied agreement of the axial component between all turbulence models and the measurement data. Only near the wall slight deviations are visible, where the measurement shows a velocity drop at the top plane. Especially in this region a higher measurement resolution would be beneficial to evaluate this behavior in near-wall region. SST and BSL-EARSM model are over-predicting the velocity close to the walls, where the \( k - \epsilon \) model exhibits a reduced axial component. The tangential velocity distributions are showing deviations of the numerical result at part load condition. Beside the simplification of turbulence anisotropy in the two-equation models which results in a reduced accuracy for swirl flows, also cavitation phenomena are not considered. However it is noticeable that the standard SST-model and the improved BSL-EARSM match especially the trend of the velocity gradient very well, yet under-predicting the swirl, where the \( k - \epsilon \) model shows a higher tangential velocity without reproducing the increasing values near the wall due to a limited behavior in rotational flows and near-wall regions. For the BSL-RSM turbulence model it is not possible to gain a numerical stable solution at this part load operating point.

At best efficiency \( k - \epsilon \), SST and BSL-EARSM show similar behavior, with increasing deviations for the axial flow near the center line and the draft tube wall (cf. figure 7). For...
the BSL-RSM approach larger discrepancies in axial velocities near the draft tube core occur. In this area the recirculation length is insufficiently predicted, resulting in increasing deviations near the draft tube center. Especially for the tangential velocity all turbulence models predict a drop near the center, indicating a rotational outflow. However, as the measurement confirms, the runner shows at the best efficiency point a nearly swirl-free outflow behavior.

![Graph](image1)

(a) top measurement plane

![Graph](image2)

(b) bottom measurement plane

Figure 7: axial (left) and circumferential (right) velocity distribution at best efficiency

Similar results in comparison with experimental data for the axial velocity can be obtained at the high load operating point. The trend of the measurements are reproduced well by all turbulence models with remaining deviations in the core zone. At high load operating conditions a counter-rotating flow at the runner exit occurs which is captured by the simulation (cf. figure 8), although the decrease in streamwise direction is over-predicted. Again only minor differences between the turbulence models are visible. In general a best fit of the axial component is found at the part load whereas at other operating points some deviations can be observed especially in the core region of the draft tube. Beside some discrepancies in the core and near wall regions and although the use of improved turbulence models show a remarkably small influence, the accordance between experimental and numerical results are satisfying. However with the focus on stable, accurate and first of all efficient solutions the BSL-RSM approach seems not...
to be appropriate for today development processes. In a further study adding and neglecting of different domains, such as spiral case and draft tube is investigated in order to indicate the significance for velocity distributions. To keep an overview only the evaluations with major influences are shown.

4.3. domain variation

According to table 2, there are four different domain modifications under investigation (G1-G4). Configuration G1 is the already discussed reference model. In a first modification the draft tube of G1 is shortened to the cone-shaped part of the draft tube (G2). A downstream extension of $5 \times D_{DT}$ is necessary to avoid influences of the outlet boundary conditions on the measurement section. In variation G3 the spiral case is added to the original domain. To circumvent an interface between stay vanes and guide vanes all 28 guide vanes are taken into account, leaving only one interface between guide vanes and runner. The last modification investigates the influence of circumferentially averaging the flow at the runner outlet on the draft tube velocity field. As shown in figure 9 the interface is shifted beneath the measurement region at the end of the conical section, allowing to evaluate the velocities in the rotating domain (G4). However, as with geometry G3, due to the curved periodical passage it is not possible to directly compare the flow at the straight measurement line. Therefore an average over the horizontal section is necessary. The results at different operating conditions are similar to previous investigations,
though two characteristics are mentionable. According to figure 10 the neglected draft tube has an influence on the tangential velocity at part load only, whereas for all other configurations no deviations compared to the reference solution are found. However a drawback of this approach is a missing statement on the hydraulic efficiency, as the draft tube losses are unavailable. At best efficiency and high load the domain variations have minor influences, where the spiral case and reference model as well as the shifted interface and the neglected draft tube show almost identical characteristics (cf. figure 11). While the velocities of configuration G\(_1\) and G\(_3\) (+SC) are evaluated in the stationary frame of the draft tube, the velocities of G\(_2\) (-DT) and G\(_4\) (DT-IF) are calculated in the rotating frame. Therefore it can be stated that a circumferentially averaging of the velocities has a major effect compared to different domain modifications.
Figure 11: velocity distribution at best efficiency for different domain models at bottom measurement plane

4.4. hydraulic efficiency

According to IEC 60193 Annex N [6] the turbine efficiency is defined as:

\[ \eta_h = \eta_E \eta_v \eta_r \]  

(7)

where due to the small dimensions of the model machine buoyancy effects can be neglected and eq. 7 results in [6]:

\[ \eta_h = \left( 1 - \frac{1}{\rho g H} \sum_{k=c} \Delta P_{tot,k} \right) \left( 1 - \frac{q' + q''}{Q} \right) - \frac{P_{Ld}}{\rho g H Q} \]  

(8)

Here c stands for the components spiral case, guide vane, wicket gate, runner and draft tube. Seal leakage losses of crown (q') and band (q'') as well as disk friction losses (P_{Ld}) are estimated from a investigation of the runner side gap (cf. chapter 4.5). Spiral case losses are chosen accordingly to the simulations in chapter 4.3 (domain variation G3) and considered for all test-cases. Figure 12 shows a comparison of the reference configuration with the measurement. All
A negative percentage value refers to an under-predicted hydraulic efficiency of the simulation compared to the measurement, whereas a positive value means an over-prediction. Model $G_2$ is excluded in this evaluation, because the missing draft tube inhibits a statement of the total hydraulic losses. According to table 4 the same tendency for all calculations is observable. At part load conditions the efficiency is under predicted up to 9% for the SST turbulence model. Deviations up to 14% at low discharge operating conditions between numerical simulations and the measurement for this model case were already discussed by Trivedi et al. [10]. For the remaining two operating points, deviations of efficiency vary between -1,4% and 0,15%. Again absolute deviations of different variations regarding model domain or turbulence model are in a very small range. The BSL-RSM approach results in best accordance with the measurements. On the contrary at part load occurring stability issues and for BEP/HL an over-predicted recirculation area in the draft tube center is found.

Table 4: deviations of hydraulic efficiency for different turbulence models and model domains compared to measurements

| OP    | PL  | BEP | HL  |
|-------|-----|-----|-----|
| absolute deviation | 71,7 | 92,4 | 91,0 |
| measurement $\eta_h$ [%] | G1-SST | -8,94 | -0,44 | -0,47 |
|       | G1-k-$\epsilon$ | -7,08 | -0,70 | -1,09 |
|       | G1-BSL-EARSM | -7,55 | -0,69 | -1,35 |
|       | G1-BSL-RSM | -0,08 | 0,15 |
|       | G3-SST | -6,59 | -0,28 | 0,01 |
|       | G4-SST | -6,94 | -0,41 | -0,15 |

4.5. runner side gap

Another area of investigation are the runner side gaps. With the input of basic geometry seal data, an in-house software is able to mesh the existing seals (cf. figure 13a). In a first approach these geometries of crown and band can be included in the numerical model for simulation (cf. figure 13b, RSG-SST in table 5). A second method considered the leakage losses with an analytical approach to calculate the seal leakage, disc friction losses as well as axial thrust based on formula from Güllich [4], Lorenz [7], Stampa [9] and Weber [12]. The resulting losses can be
taken into account in the loss analysis for the hydraulic efficiency according to eq. 8 as done in all previous simulations (cf. table 4). For the hydraulic efficiency it is possible to improve the prediction as shown in table 5. As a result in this case the seal leakage and disc friction losses are over-predicted in the analytical approach. However a consideration of the three-dimensional seals in the numerical model leads to very small mesh sizes in streamwise direction, resulting in stability issues especially at operating points apart from best efficiency. Therefore it can be stated that if the seals are not under special investigation it is sufficient, especially in terms of numerical stability and efficiency, to consider the seal leakage in an analytical way and take it into account for the resulting hydraulic efficiency.

Table 5: deviations of hydraulic efficiency for a runner side gap consideration compared to measurements

| OP absolute deviation | PL  | BEP | HL  |
|-----------------------|-----|-----|-----|
| measurement $\eta_h$ [%] | 71.7 | 92.4 | 91.0 |
| $G_1$-SST             | -8.94 | -0.44 | -0.47 |
| RSG-SST               | -6.45 | 0.00  | -0.20 |

4.6. optimized model

Since there are no relevant improvements by using higher sophisticated turbulence models, at least not in a order to justify the increasing complexity, numerical instability and computational time, the $k – \omega - SST$ turbulence model is used for the following configuration. Furthermore the spiral case has no significant influence on the velocity distribution, whereas modeling the draft tube is vital, especially at part load conditions. Therefore the domain variation under investigation is $G_1$ with the refined mesh $M_3$. Although the total mesh size is tripled, the improvements vary within a very small range. At part load no change of the velocity distribution is observable, whereas at best efficiency (cf. figure 14) and high load a slightly shift towards the measurement occurs. The axial velocity field remains similar, while the appearing swirl near the runner hub is decreasing. With a higher mesh density better predictions of the hydraulic

![Figure 14: velocity distribution at best efficiency with a refined mesh at bottom measurement plane](image-url)
losses are possible. This results in a reduced hydraulic efficiency which in this case, as the efficiency is already underrated for BEP and HL, leads to a greater deviation in comparison with measurement data (cf. table 6).

Table 6: deviations of hydraulic efficiency for a refined mesh model compared to measurements

| OP absolute deviation | PL | BEP | HL |
|-----------------------|----|-----|----|
| measurement $\eta_h$ [%] | 71,7 | 92,4 | 91,0 |
| G1-M2-SST | -7,98 | -0,44 | -0,47 |
| G1-M3-SST | -5,89 | -0,82 | -0,65 |

5. Discussion
Increasing computational resources allow the use of more advanced turbulence models as well as refined grid densities and the simulation within reasonable time spans. A high head Francis model turbine is modeled for one passage of distributor and runner, applying periodical boundary conditions, with a full draft tube. Three operating points at part load, best efficiency and high load are under investigation. With the in-house procedure a fast, reliable meshing as well as setup-procedure and post-processing is possible. Pressures at six locations in the rotating and stationary frame are analyzed and differences are found to $\pm 5\%$ in the runner, $-7\%$ in the draft tube and up to $-15\%$ in the vaneless space when compared to the experimental data. The application of different two-equation and Reynolds stress turbulence models showed minor influences on the velocity distribution in the draft tube beneath the runner. However with the BSL-RSM turbulence model the recirculation area beneath the runner is over-predicted which led to deviations for the velocity field in the draft tube in comparison with other turbulence models. Overall the $k-\omega-SST$ turbulence model confirmed the best solution in terms of robustness, stability and accuracy. Domain variations indicated an influence of the draft tube at part load conditions on the occurring swirl, where at other operating points the interface position at which a circumferential averaging of the flow variables is carried out showed the major effect. Hydraulic efficiency is well predicted at the operating points BEP and HL with a tendency to smaller values, where at PL larger deviations occurred, which were also discussed by other authors. The included runner side gaps improved the prediction of the hydraulic efficiency with the drawbacks of increasing computational time and reduced numerical stability. A refined mesh reduced discrepancies of the tangential velocity at best efficiency while the increasing losses resulted in larger deviation of the hydraulic efficiency. Although the results showed good agreement in terms of velocity and hydraulic efficiency, further investigations for efficiencies at part load conditions and velocity distribution at best efficiency operating point are necessary. Further mesh refinements or the detailed trailing edge meshing will influence the over-predicted swirl at best efficiency. Also the use of scale resolving turbulence models as large eddy simulations (LES) could improve the simulation results.

Nomenclature

| Term                       | Symbol | Unit        | Indices |
|----------------------------|--------|-------------|---------|
| Velocity, absolute         | $c$    | m/s         | 0       |
| Diameter                   | $D$    | m           | 1       |
| Gravity acceleration       | $g$    | m/s$^2$     | 2       |
| Hydraulic head             | $H$    | m           | ax      |
| Turbulent kinetic energy   | $k$    | m$^2$/s$^2$ | eul     |
| Term                        | Symbol | Unit   | Indices   |
|-----------------------------|--------|--------|-----------|
| Runner angular speed        | n      | 1/s    | exp       |
| Speed factor                | \( n_{ed} \) | -      | E         |
| Pressure                    | p      | Pa     | in        |
| Disk friction losses        | \( P_{Ld} \) | W      | h         |
| Leakage losses              | \( Q, q' \) | m³/s   | ref       |
| Discharge                   | \( Q_{ed} \) | -      | u         |
| Residuum                    | r      | -      | v         |
| Mean strain rate tensor     | \( S_{ij} \) | 1/s    |           |
| Time                        | t      | s      |           |
| Velocity, components in CCS | \( u_i, u_j \) | m/s | CCS       |
| Velocity, circumferential   | \( u \) | m/s   | CFD       |
| Dimensionless wall distance | \( y^+ \) | -      | BEP       |
| Guide vane angle            | \( \gamma \) | °      | DT        |
| Kronecker delta             | \( \delta_{ij} \) | -     | EARSMD    |
| Difference                  | \( \Delta \) | -      | G1-G4     |
| Efficiency                  | \( \eta \) | %      | IF        |
| Turbulence eddy viscosity   | \( \nu_T \) | m²/s  | LDA       |
| Density                     | \( \rho \) | kg/m³ | M1-M3     |

**Abbreviations**

- BSL: Baseline turbulence model
- CCS: Cartesian coordinate system
- CFD: Computational fluid dynamics
- BEP: Best efficiency
- DT: Draft tube
- EARSMD: Explicit algebraic RSM
- G1-G4: Domain variation
- IF: Interface
- LDA: Laser Doppler Anemometer
- M1-M3: Mesh variation
- PL: Part load
- RSG: Runner side gap
- RSM: Reynolds stress model
- RU: Runner
- SC: Spiral case
- SST: Shear stress turbulence model
- SV: Stay vanes
- WG: Guide vanes

**References**

[1] ANSYS Inc: *ANSYS CFX-Solver Theory Guide*, release 15.0, 2011
[2] Aschenbrenner, T.; Otto, A.; Moser, W., *Classification of vortex and cavitation phenomena and assessment of CFD prediction capabilities*, 23rd IAHR Symposium on Hydraulic Machinery and Systems, 2006
[3] Cervantes, M.: *Francis-99 Workshop*, Lulea University of Technology, http://www.ltu.se/research/subjects/Stromningslara/Konferenser/Francis-99, accessed August 15, 2014
[4] Güllich, J.F.: *Disk friction losses of closed turbomachine impellers*, Forschung im Ingenieurwesen, vol. 68, p. 87-95, 2003
[5] Hoffmann, K.; Chiang S.: *Computational Fluid Dynamics*, vol. 3, Engineering Education System, 2000
[6] International Electrotechnical Comission; *IEC 60193: Hydraulic turbines, storage pumps and pump turbines - Model acceptance test*, International Electrotechnical Commission, Geneva, Switzerland, 1999
[7] Lorenz, J.: *Untersuchung von Spaltdichtungen für hydraulische Strömungsmaschinen*, Master-Thesis, University of Konstanz 1986
[8] Menter, F.: *Improved two equation k-\( \omega \) turbulence model for aerodynamic flows*, NASA technical memorandum 103975, 1992
[9] Stampa, B.: *Experimentelle Untersuchung an axial durchströmten Ringspalten*, PhD-Thesis, Technische Universität Braunschweig, 1971
[10] Trivedi, C.; Cervantes, J.M.; Ghandi, B.K.; Dahlhaug, O.G.: *Experimental and numerical studies for a high head Francis turbine at several operating points*, Journal of Fluid Engineering, vol. 135, 2013
[11] Wallin, S; Johansson A.: *An explicit algebraic Reynolds stress model for incompressible and compressible turbulent flows*, Journal of Fluid Mechanics, vol. 403, pp. 89-132, 2000
[12] Weber, D.: *Experimentelle Untersuchung an axial durchströmten kreisringförmigen Spaltdichtungen für Kreiselpumpen*, PhD-Thesis, Technische Universität Braunschweig, 1971
[13] Wilcox, D.: *Turbulence Modelling for CFD*, DCW Industries, Inc., 1994