Predicting the performance of a high head Francis turbine using a fully implicit mixing plane

O Amstutz, B Aakti, E Casartelli, L Mangani, L Hanemann
Lucerne University of Applied Sciences and Arts Engineering & Architecture, Horw, Switzerland
E-mail: oliver.amstutz@hslu.ch

Abstract. In the present paper numerical investigations of a complete high head Francis turbine comprehensive of a spiral casing, stay and guide vanes and draft tube have been performed at three operating conditions, namely at part load (PL), best efficiency point (BEP), and high load (HL). The main target of the investigations is to assess the prediction accuracy of a reduced domain of the complete turbine using a novel mixing-plane formulation.

The computational domain is simplified simulating one single passage of the runner, thus assuming rotational periodicity and steady state conditions. The results were compared with experimental data published by the workshop organization.

All CFD simulations were performed at model scale with an in-house adapted, 3D, unstructured, object-oriented finite volume code based on the OpenFOAM-V2.2 framework and designed to solve steady-state incompressible RANS-Equations. The pressure-based solver uses a SIMPLE-C like algorithm and is capable of handling multiple references of frame (MRF). The influence of the turbulence has been considered applying the shear-stress transport model (SST). Full second order upwind scheme for advection discretization has been used for all computations.

1. Introduction

The scope of the first Francis-99 workshop is the assessment of advanced numerical tools for the prediction of steady state conditions in a Francis turbine for design and off-design conditions [1]. The continuous evolution of the numerical techniques and the computer power has lead in the last decade to a large tool-spectrum for turbomachinery investigations. The driver for this trend was mainly the only partly-satisfactory results obtained, especially at off-design conditions. The main reasons for the discrepancies were located firstly in the turbulence models. With the development of more advanced models capable to improve the well-known drawbacks of classical models like $k-\epsilon$ and capture additional flow physics, like curvature correction for the SST model or the v2F model. Secondly they were located in the thought inability of steady state methods to properly predict the inherently unsteady effects present in turbomachines [2], [3].

A detailed analysis of one of the first methods to efficiently investigate the interaction of turbomachinery blade-rows, the mixing plane method [4], has shown that the usual implementation can be improved using a fully implicit approach, compared to the until now used explicit one. The main advantage are (1) an increased stability and robustness, allowing for example to handle backflow at the interface, as well as an increased accuracy, allowing to predict not only globally but also locally reliable distributions of the flow quantities in a time-averaged, i.e. steady-state way over a large operating range [5].

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
2. CFD - solver technology

2.1. Introduction and concept
The design of modern multi-stage turbomachines is a challenging task for engineers, which can only be tackled with the help of CFD. While in the last few years new approaches based on unsteady methods have been developed for single-passage computations of the rotor-stator interaction, thus promising to capture unsteady effects already in the design procedure though not computing the full annulus [2, 3], the steady-state approach still represents the most used industrial standard [4, 6, 7].

Of the steady-state methods, the mixing-plane approach is the most common and general, assuming, as observed in many real turbomachinery applications, that the flow largely mixes out in the gap between two consecutive blade rows (figure 1). This leads to the assumption that at the interface between the components pitch-wise constant total conditions have to be imposed. Therefore a mixing plane mimics the time averaged impact of rotor-stator interactions over a complete revolution.

![Figure 1: Concept of mixing plane averaging.](image)

Despite the similarity between the mixing process and the mixed-out state, the difference in the way the mixing happens can cause problems. Even if the flow is largely mixed out in reality, the process is more gradual than the instantaneous mixing at the mixing plane. This instantaneous mixing will result in a higher entropy rise and can be seen as a loss of total pressure over the interface. Studies by Fritsch and Giles [8] showed an increase in total loss of about 10% when instantaneous mixing was applied.

2.2. Problems of standard procedure
The usual methodology for the mixing plane implementation is based on an explicit treatment of the interface. This leads to a numerical separation of the problem which has to be coupled with a new boundary-condition set between the two components, constantly exchanging averaged primitive variables. While the approach is simple in the idea, numerical problems can occur by the division of the computational domain in two parts.

A number of authors have investigated the influence of specific averaging methods, agreeing that simple area or mass averaging can not guarantee conservation of mass, momentum and energy. Dzung’s [9] consistent averaging is one possible option. Another advanced method using all relevant conservation equations and making the distinction between hypothetically reversible and irreversible quantities was presented in a publication by Kreitmeier [10].
Amongst the difficulty to fulfill conservation equations during the averaging procedure of flow variables (see [9, 10]), one of the most discussed issues are reflections of information at the interface boundaries.

The strong constraint of pitch-wise uniform distribution is accompanied by the problem of reflection of information, as ordinary in- and outlets have. However, for the mixing plane approach these reflections can not be fully assigned to an inappropriate modeling of the farfield at the boundaries of a truncated domain. It is mainly caused by the very basic model-assumption of uniform total conditions in pitch-wise direction. Additionally it has to be mentioned that reflections produce wiggles in the solution, which can cause difficulties in the numerical procedure, deteriorating convergence and thus leading to longer computational time.

As a counter-measure a lot of effort in commercial and proprietary codes was and still is undertaken in the community, for the development of so-called non-reflecting boundary conditions (NRBC), considered a must for a mixing-plane interface in order to work properly (see for example [7], [11], [12]). NRBC are derived from the general unsteady approach, where waves with different frequency have to pass the interface between rotor and stator without being reflected, thus defining appropriate boundary conditions in order to vanish incoming disturbances.

The mixing-plane is thus usually defined in two steps as given in [13]: first, an average state is imposed corresponding to the zero frequency of the wave representing the steady state situation and second, local perturbations are superimposed in order to achieve a non-reflective behavior, based on the Fourier analysis of linearized Euler’s equation for the corresponding boundary condition. Out of this approach a series of problems arises in order to achieve both consistent averaging and NRBC behavior and are directly linked to the explicit treatment of the interface. Accordingly, the authors published in [14] a novel approach for the implementation of the mixing-plane, based on a fully implicit treatment of the interface, giving details on the implementation and validating the interface with measurements. The most important fact of the implicit discretization is that it takes directly into account both consistent averaging and reflectivity, as explained in the section "Implicit Mixing Plane".

2.3. New Formulation

A simplified example will be given to point out the main difference between the standard explicit treatment and the new implicit formulation. Figure 2 represents a simplified domain, consisting of four cells and separated by a mixing plane.

![Figure 2: Example domain.](image)

The differences in the coupling of the interface will be pointed out by a numerical discretization of a diffusion operator, given in equation 1.

\[
B = \int \nabla_{cell} \Phi dV = \int \Gamma_{cell} \Phi dS \approx \sum \Gamma_f \Phi_f S_f
\]  

(1)

This equation can be rewritten in the general form shown in equation 2.
\[
A \cdot x = b
\] (2)

For further explanations the fluxes across the faces will be discretized with a central difference scheme and the areas of the faces are of uniform size \( S_f \). Furthermore \( \Gamma = \Gamma_f \cdot S_f \).

**Explicit Algorithm**

As mentioned earlier an explicit coupling leads to additional in- and outlets as visualized in figure 3.

![Figure 3: Explicit separation.](image)

The sub-domains are represented by two systems of linearized equations and given in equation 3 and 4.

\[
\begin{bmatrix}
\Gamma & \Gamma & \Gamma & \Gamma
\
\Phi_1 & \Phi_2 & \Phi_3 & \Phi_4
\end{bmatrix}
\begin{bmatrix}
A_1
\end{bmatrix}
\begin{bmatrix}
x_1
b_1
\end{bmatrix}
= 
\begin{bmatrix}
\Gamma & \Gamma & \Gamma & \Gamma
\
\Phi_3 & \Phi_4 & \Phi_1 & \Phi_2
\end{bmatrix}
\begin{bmatrix}
x_2
b_2
\end{bmatrix}
\]
(3)

\[
\begin{bmatrix}
\Gamma & \Gamma & \Gamma & \Gamma
\
\Phi_3 & \Phi_4 & \Phi_1 & \Phi_2
\end{bmatrix}
\begin{bmatrix}
x_2
b_2
\end{bmatrix}
= 
\begin{bmatrix}
\Gamma & \Gamma & \Gamma & \Gamma
\
\Phi_1 & \Phi_2 & \Phi_3 & \Phi_4
\end{bmatrix}
\begin{bmatrix}
x_1
b_1
\end{bmatrix}
\]
(4)

The values from the adjacent side of the interface are used as inlet and outlet conditions according to the theory of characteristics. The influence of the adjacent domain fully acts as an explicit contribution to the source. Therefore, the coupling is only carried out before the outer iterations.

**Implicit Algorithm**

An implicit treatment of a coupled interface can be understood as using two separate matrix of coefficients to build up a system of linearized equations for the complete solution domain as shown in equation 5

\[
b = A \cdot x = (A^I + A^C) \cdot x
\] (5)

The interface is no longer build up from an additional in- and outlet (figure 4), instead the fluxes over the interface are integrated into the system of linearized equations.

![Figure 4: Implicit separation.](image)
The system of linearized equations for the inner coefficients is build up as follows.

\[
\begin{bmatrix}
\Gamma_2 & \Gamma_2 & 0 & 0 \\
\Gamma_2 & \Gamma_2 & 0 & 0 \\
0 & 0 & \Gamma_2 & \Gamma_2 \\
0 & 0 & \Gamma_2 & \Gamma_2 \\
\end{bmatrix} \times \begin{bmatrix}
\Phi_1 \\
\Phi_2 \\
\Phi_3 \\
\Phi_4 \\
\end{bmatrix} = \begin{bmatrix}
b_1 \\
b_2 \\
b_3 \\
b_4 \\
\end{bmatrix}
\] (6)

It can be seen, that the system shown in equation 6 can be build up from the equation systems of the explicit approach. However, instead of considering the influence of the coupling in the source term, these fluxes are implicitly discretized. A system of linearized equations for the interface fluxes, respecting the above given example is shown in equation 7.

\[
\begin{bmatrix}
0 & 0 & 0 & 0 \\
0 & \Gamma_2 & \Gamma_2 & 0 \\
0 & \Gamma_2 & \Gamma_2 & 0 \\
0 & 0 & 0 & 0 \\
\end{bmatrix} \times \begin{bmatrix}
\Phi_1 \\
\Phi_2 \\
\Phi_3 \\
\Phi_4 \\
\end{bmatrix} = \begin{bmatrix}
b_1 \\
b_2 \\
b_3 \\
b_4 \\
\end{bmatrix}
\] (7)

Adding $A^I + A^C$ will lead to a system of linearized equations for the complete domain.

\[
\begin{bmatrix}
\Gamma_2 & \Gamma_2 & 0 & 0 \\
\Gamma_2 & \Gamma_2 & 0 & 0 \\
0 & \Gamma_2 & \Gamma_2 & 0 \\
0 & 0 & \Gamma_2 & \Gamma_2 \\
\end{bmatrix} \times \begin{bmatrix}
\Phi_1 \\
\Phi_2 \\
\Phi_3 \\
\Phi_4 \\
\end{bmatrix} = \begin{bmatrix}
b_1 \\
b_2 \\
b_3 \\
b_4 \\
\end{bmatrix}
\] (8)

The coefficients of the linearized equations for the owner cells to their faces are always local while the coefficients from the faces to the coupled cells are averaged coefficients from the adjacent side. With this implicit treatment, the iterative solution procedure constantly considers the complete domain, even during the inner iterations.

Defining the boundary implicitly, i.e. as a boundary which is fully coupled within the domain like for instance periodic boundaries, no additional numerical treatment is needed to avoid numerical artifacts. The key point is that this approach just defines the relation of the boundary interaction directly in the discretization of the domain, without forcing any constraint through an additional BC. This allows to have a fully consistent system based on the accuracy and the physics of the equations solved in the domain. Please note that there are already other implicit boundaries used in CFD, like for instance periodicities. Here no additional non-reflective treatment is needed: there is only an implicit relationship between the two sides. This fact is actually often used in LES and DNS, known as very sensitive regarding the numerics. In this kind of simulations, imposing for example an inlet BC can easily lead to a non-physical flow field, since all the necessary quantities can usually not be given in a consistent way (i.e. divergence free) for all time and length scales resolved.

3. Case setup
3.1. Geometry
In figure 5 the complete model is depicted, with the various components in different colors. As listed in Table 1, the turbine has 15 main and 15 splitter blades, 28 guide and 14 stay vanes. For the three operating points different guide-vane blade-angle are defined: 9.84 degree for the
BEP, 3.91 degree for PL and 12.44 degree for the HL case. Further geometrical details of the model are given in [15], [16].

![Diagram of the Francis-99 turbine simulation domains](image)

**Figure 5:** The four simulation domains of the Francis-99 turbine.

**Table 1:** Main geometry data of the Francis-99 turbine.

| Blade No (splitter blades) | GV blade No | SV blade No | Max diameter runner [mm] | Max diameter GV [mm] | Max diameter SV [mm] |
|---------------------------|-------------|-------------|--------------------------|----------------------|----------------------|
| 15 (+15)                  | 28          | 14          | 631                      | 764                  | 980                  |

### 3.2. Model simplifications
In order to reduce the complexity of the simulations, several model simplifications have been adopted. The clearance gap at the guide vanes are neglected. The model is assumed to behave as a rigid body which is experiencing no deformations. Moreover, the rotor side spaces were neglected so that disc friction as well as leakage flow were not directly available in the simulations. In order to estimate the overall effect of the neglected leakage within the rotor side spaces, an in-house tool [17] has been used to determine the friction torque in the leakage gap. The in-house tool uses interpolation equations which, from an extensive dataset based on detailed parameter studies of rotor side spaces, extract the corresponding correlation values [18]. The calculation is limited to the leakage gap on the shroud side of the runner because on the hub side of the runner there are no pressure relief-holes and therefore the influence is considered as not relevant. At BEP conditions, the friction torque within the leakage gap is 9.53 Nm which corresponds to 1.8% of the measured blade torque. At HL and PL, the friction torque is 1.5% and 8.0%, respectively. These values are used to correct the measured efficiency, thus leading to higher values, which are then in good agreement with the computed efficiency.

### 3.3. Mesh
In addition to the available geometry and mesh data from the Francis-99 workshop, used in [15], an improved structured mesh, named hereafter HSLU-Mesh, has been created. Particularly, the boundary layer around the blades, splitter blades, guide vanes and stay vanes has been refined. The HSLU-Mesh consists in total of 11 million elements and the average $Y^+$ value is 22 at BEP. Table 2 summarizes the detailed mesh quality of the individual computation domains.
Table 2: HSLU-Mesh quality.

| HSLU-Mesh          | Element size | $Y^+$ (BEP) | Angle | Volume change | Aspect ratio |
|-------------------|--------------|-------------|-------|---------------|--------------|
| Global            | 10'995'677   | 21.8        | 2.8   | 97            | 1421         |
| Spiral & stay vanes | 5'331'788    | 19.1        | 3.5   | 50            | 1421         |
| Guide vanes BEP   | 3'780'280    | 26.9        | 19.5  | 56            | 58           |
| Guide vanes PL    | 3'780'280    | 19.4        | 17.2  | 97            | 153          |
| Guide vanes HL    | 3'780'280    | 23.6        | 16.0  | 55            | 73           |
| Runner            | 398'930      | 15.7        | 2.8   | 21            | 305          |
| Draft tube        | 1'484'679    | 21.8        | 51.8  | 8             | 250          |

3.4. Comparison between LTU-Mesh and HSLU-Mesh

In Table 3 is the mesh data of the provided LTU-Mesh. A main difference between these meshes is the value of the average $Y^+$ which is 120 and 22 for the LTU-Mesh and the HSLU-Mesh, respectively. The LTU-Mesh consists of 7.5 million elements and is therefore by 32 percent smaller than the HSLU-Mesh in terms of the number of elements. In order to achieve a better mesh quality, the mesh domain of the guide and stay vanes has been separated into two domains.

Table 3: LTU-Mesh quality.

| LTU-Mesh          | Element size | $Y^+$ (BEP) | Angle | Volume change | Aspect ratio |
|-------------------|--------------|-------------|-------|---------------|--------------|
| Global            | 7'513'862    | 119.7       | 2.8   | 164           | 52           |
| Spiral, SV, GV BEP | 3'607'016    | 182.4       | 12    | 164           | 52           |
| Spiral, SV, GV PL | 3'441'063    | 89.6        | 7.7   | 81            | 49           |
| Spiral, SV, GV HL | 3'581'240    | 170.2       | 7.7   | 43            | 46           |
| Runner            | 335'902      | 106.0       | 2.8   | 54            | 47           |
| Draft tube        | 3'570'944    | 26.3        | 50.7  | 3             | 38           |

The resulting torque of both meshes is similar, but the LTU-Mesh shows less losses and therefore a overestimation of the net head. Figure 6 visualize the flow around a guide vane. The boundary layers along the walls as well as the wake flow behind the trailing edge is not (well) resolved by the LTU-Mesh being too coarse to capture these phenomena. This could be the main reason for the overestimation of the net head.

(a) LTU-Mesh

(b) HSLU-Mesh

Figure 6: Guide vanes with the same total pressure contour plot scale.
3.5. Setup
At the inlet of the computational domain the velocity \( u_{\text{inlet}} \), the turbulent kinetic energy (\( k \)), and the turbulent frequency (\( \omega \)) are imposed as a fixed value. The turbulence variables \( k \), \( \epsilon \) and \( \omega \) are initialized as \( k = \frac{3}{2} \left| u_{\text{inlet}} \right|^2 \), \( \epsilon = C_\mu^{0.75} k^{1.5} / \ell \) and \( \omega = \epsilon / k / C_\mu \) respectively, where \( I \) is the prescribed turbulence intensity of 0.05 respectively 5%, \( C_\mu = 0.09 \) is a constant turbulence model parameter and \( \ell \) is the turbulent mixing length scale. At the outlet of the computational domain, an average static pressure of \( p_{\text{outlet}} = 0 \) Pa is used. The walls in the domains are non-rotating, except for the runner (hub, blade and shroud). At the walls, a no-slip condition is imposed. The runner domain rotates around the negative z-axis.

Figure 5 shows the individual components of the computational domain. The rotational periodicity of the runner passage is modeled applying a one-to-one interface. Between the guide vanes and the runner as well as between the runner and the draft tube, the implicit HSLU mixing plane interface is applied. Between the stay vanes and the guide vanes an AMI-Interface is placed.

All the simulations were performed using the \( k-\omega - \text{SST} \) turbulence model which requires the numerical solution of two scalar equations for the turbulent conservation variables in addition to momentum and continuity equations. Although two-equation turbulence models are known to have shortcomings at off-design conditions in the past, even flow instabilities were successfully investigated, as shown in [19, 20], where they were captured with fairly good agreement compared to experimental data. The choice of the \( \text{SST} \) turbulence model is therefore based on the fact that at present it is regarded as a sort of industrial standard and widely used [21, 22, 23, 24] for this kind of simulation.

All CFD simulations were performed at model-scale with an in-house as well as with a commercial code, solving the steady-state incompressible RANS-Equations in double precision with a full second order upwind scheme for advection discretization.

The in-house code is an adapted, 3D, unstructured, object-oriented finite volume code based on the OpenFOAM-V2.2 framework. The segregated pressure-based solver uses a SIMPLE-C like algorithm and is capable of handling multiple references of frame [25]. The reference commercial finite volume solver is a pressure based coupled solver. All configurations have a mixing plane interface between the stationary and rotating domain.

4. Results
4.1. Integral values
Figure 7 compares the efficiency, the net head, and the torque around the z-axis, at all given operating points. The comparison leads to the following findings:

- The efficiency in the simulations with the HSLU mixing plane is in all three operating points closer to the measurement than that of the commercial code (CC).
- Except for the PL case, the same behavior results for the differential pressure and the net head, respectively.
- The torque of the runner around the z-axis is lower than the simulated torque by the commercial software and is therefore closer to the experiment.

From a comparison with 360° unsteady results [26] it can be stated that no or less influence of unsteadiness exists in BEP and HL operating condition. Since the global results are very similar to the one of the mixing plane. However at part load operating point the rotor-stator interaction seems to be important. The mixing plane underestimates the efficiency because of the neglected unsteady effects. For example in part load condition, the flow in every channel in 360° can be different [26]. The main reason for the better accuracy of the OpenFOAM (OF) simulations compared to the commercial-software results is due, in the opinion of the present authors, to the fully implicit treatment of the mixing plane which imposes a circumferentially
uniform distribution of the total pressure immediately downstream of the interface. Denton [4] states that a mixing plane should give a solution which is independent of the relative position of rotor and stator, resulting in uniform pitch-wise enthalpy and entropy for a fixed span-wise height on the downstream side of the domain, in order to produce a better representation of the average effect of the rotor-stator interactions. Defining the boundary implicitly, i.e. as a boundary which is fully coupled within the domain like for instance periodic boundaries, no additional numerical treatment is needed to avoid numerical artifacts. This leads to considerable advantages compared to the state-of-the-art mixing plane: (1) the solution fully satisfies the mixing plane constraint of uniform total conditions in pitch-wise direction, while (2) the continuity of the solution reduces incoming disturbances having therefore a sort of built-in non-reflectiveness. At the same time (3) improved robustness is also demonstrated with the ability to handle backflow at the interface. This leads to a consistent prediction of the blade loading and therefore to more accurate results.

Figure 7: Integral values in all three operating points.

Figure 8 shows exemplary the difference between the commercial software and OpenFOAM at the downstream mixing-plane for the BEP conditions. The figures show the total pressure in stationary frame with the same scale, respectively. It can be seen that at the draft tube mixing-plane the HSLU solution is completely axisymmetric, i.e. fully repeating the definition of the mixing plane approach.

Figure 8: Total pressure at the mixing plane location between the runner passage and the draft tube. The values of the contour lines are the same in both figures.
Figure 9 shows an axial cut of runner and draft-tube at part load. At this condition the flow through the machine is concentrated along the walls of the draft-tube, while in the center a large back-flow region is present. Here a further strength of the fully implicit approach can be seen: the back flow at the interface can be handled without numerical drawbacks, i.e. not affecting the convergence behavior. The authors believe that both circumferentially uniform total-pressure and the ability to handle back-flow at the interface are the reasons for the good integral performance of the steady-state one-passage approach compared to experiment and 360° data [26].

4.2. LDA Measurements
In the following, the mean axial and tangential velocity profiles along the radius at two measuring sections, as described in figure 10, are compared. In order to ensure a good comparability of the CFD and the Laser Doppler Anemometry (LDA) results, the velocities predicted by CFD have been evaluated exactly at the same position as the LDA measurements.

The performance of the two CFD codes regarding the prediction-accuracy of the velocity components is similar at off-design conditions for the top and the bottom slice. Therefore, only the results at the top slice are given in figures 11 and 13 for PL and HL.

At PL, both CFD codes predict similar results for the axial and tangential velocity along the radius. The axial velocity is in both codes slightly overestimated compared to the measurement.
At design conditions (figure 12), both CFD codes are fairly close to the experimental results of the axial velocity. OpenFOAM is generally more accurate predicting the tangential velocity. However, both codes clearly overestimates the tangential velocity close to the center. Nevertheless the area involved is small. A discussion of the discrepancy between the numerically predicted tangential velocities and the experimental results is given in the paper by Aakti et al. [26].

At HL, both CFD codes underestimate the axial velocity close to the center, while the commercial code is closer to the experiments. The tangential velocity is overestimated by both CFD codes over the entire radius.
5. Conclusion

Steady-state CFD computations of a high-head Francis-turbine model have been performed considering only one runner passage. At the rotor-stator interfaces, a novel fully implicit mixing plane has been applied.

The new mixing-plane formulation has been implemented in an inhouse modified OpenFOAM version. The prediction of the overall performance (efficiency, net head and torque) is fairly accurate compared to experiments and to a reference commercial code. However, both CFD codes overestimated the measured values.

Regarding the local flow distributions at runner outlet, the comparison with the measurements is satisfactory except for the tangential velocity in the BEP condition, showing that the proposed mixing-plane approach is also able to capture the local flow physics consistently over a wide operating range.

References

[1] NTNU and LTU Francis-99 workshop http://www.ltu.se/research/subjects/Stromningslara/Konferenser/Francis-99?l=en accessed August 2014
[2] Biesinger T, Cornelius C, Rube C, Braune A, Campregher R, Godin P G, Schmid G and Zori L 2010 Unsteady CFD methods in a commercial solver for turbomachinery applications ASME Conference Proceedings 2010 44021
[3] Biesinger T, Cornelius C, Nuernberger D and Rube C 2012 Speed line computation of a transonic compressor stage with unsteady cfd methods ASME Turbo Expo 2012 68029
[4] Denton J 1992 The calculation of three-dimensional viscous flow through multistage turbomachines Journal of Turbomachinery 114 p 18-26
[5] Hanimann L, Mangani L, Casartelli E, Mokulys T and Mauri S 2013 Development of a novel mixing plane interface using a fully implicit averaging for stage analysis Journal of Turbomachinery 136(8) 081010
[6] Denton J D 2010 Some limitations of turbomachinery CFD ASME Conference Proceedings 2010 44021
[7] Holmes D G 2008 Mixing planes revisited: A steady mixing plane approach designed to combine high levels of conservation and robustness ASME Conference Proceedings 2008 4361
[8] Fritsch G and Giles M 1993 An asymptotic analysis of mixing loss Journal of Turbomachinery 117(3) p 367-374
[9] Dzung L 1971 Konsistente mittelwerte in der theorie der turbomaschinen fuer kompressible medien Brown-Boveri Mitteilungen 10-71 p 485-492
[10] Kreitmeier F 1992 Space-averaging 3d flows using strictly formulated balance equations in turbomachinery ASME COGEN-TURBO 7 p 397-408
[11] Anker J E, Schrader B, Seybold U, Mayer J F and Casey M V A three-dimensional non-reflecting boundary condition treatment for steady-state flow simulation 44th AIAA Aerospace Sciences Meeting and Exhibit 44 1275
[12] Robens S, Jeschke P, Frey C, Kügeler E, Bosco A and Breuer T 2013 Adaptation of giles non-local non-reflecting boundary conditions for a cell-centered solver for turbomachinery applications ASME Turbo Expo GT2013 94957
[13] Giles M 1988 non-reflecting boundary conditions for the Euler equation TR 88 1
[14] Hanimann L, Mangani L, Casartelli E, Molukys T and Mauri S 2013 Development of a novel mixing plane interface using a fully implicit averaging for stage analysis in an object-oriented CFD-code ASME Turbo Expo GT2013 94390
[15] Trivedi C, Cervantes M J, Gandhi B K and Dahlhaug O G 2013 Experimental and numerical studies for a high head francis turbine at several operating points Journal of Flows Engineering 135(11) 111102
[16] NTNU and LTU Francis-99 test case http://www.ltu.se/research/subjects/Stromningslara/Konferenser/Francis-99/Test-Case-1.111520?l=en accessed august 2014
[17] Staubli T, Loetscher B and Cimmino D 2005 Integrale laufrad- simulationen von hydromaschinen Tech. rep. FHZ-HTA
[18] Amoser M 1995 Stroemungsfelder und Radialkraefte in Labyrinthdichtungen hydraulischer Stromungsmaschinen Ph.D. thesis ETH Zuerich
[19] Widmer C, Staubli T and Ledergerber N 2011 Unstable characteristics and rotating stall in turbine brake operation of pump-turbines Journal of Fluids Engineering 133 041101
[20] Yan J P, Seidel U and Koutnik J 2012 Numerical simulation of hydrodynamics in a pump-turbine at off-design operating conditions in turbine mode 26th IAHR Symposium on Hydraulic Machinery and Systems 2014 2.2.1
[21] Feng J J, Li W F, Wu H, Lu J L, Liao W L and Luo X Q 2014 Investigation on pressure fluctuation in a francis turbine with improvement measures 27th IAHR Symposium on Hydraulic Machinery and Systems 2014 2.2.6
[22] Tamura Y, Tani K and Okamoto N 2014 Experimental and numerical investigation of unsteady behavior of cavitating vortices in draft tube of low specific speed francis turbine 27th IAHR Symposium on Hydraulic Machinery and Systems 2014 3.1.6
[23] Heo M W, Kim K Y, Ma S B, Yoo I S, Choi W C, Kim J H and Choi Y S 2014 Parametric performance evaluation of a hydraulic centrifugal pump 27th IAHR Symposium on Hydraulic Machinery and Systems 2014 3.1.6
[24] Guo P C, Guo D M, Sun L G and Luo X Q 2014 Numerical investigation of different volutes in a low specific speed centrifugal pump 27th IAHR Symposium on Hydraulic Machinery and Systems 2014 3.1.6
[25] Mangani L, Casartelli E and Mauri S 2011 Assessment of various turbulence models in a high pressure ratio centrifugal compressor with an object oriented CFD code ASME Conference Proceedings 2011 54679
[26] Aakti B, Amstutz O, Casartelli E, Romanelli G and Mangani L 2014 On the performance of a high head francis turbine at design and off-design conditions Francis-99 workshop 1