

Assessment of an implicit mixing plane approach for pump-turbine applications

L Mangani¹, E Casartelli¹, L Hanimann¹, M Wild², N Spyrou²

¹Lucerne University of Applied Sciences & Arts, Department of Mechanical Engineering, Technikumstrasse 21, 6048, Luzern, CH

²ANDRITZ HYDRO AG, 8021 Zürich, Switzerland

E-mail: ernesto.casartelli@hslu.ch

Abstract. In the design process of pump-turbines, both in pump and turbine mode, the assessment of the components matching is very important. In order to be fast in this task, the usual procedure is based on steady-state methods, like the frozen-rotor or the mixing-plane method.

The frozen-rotor approach is straight forward and relatively easy to implement, but can produce unphysical behavior, mainly depending on the relative position of the components. On the other hand, the mixing-plane has a more physical background, delivering to the downstream component the mixed-out state of the upstream flow. On how the mixed-out state is computed and imposed to the downstream component there are different methodologies.

In the present paper a novel, fully-implicit mixing-plane method is presented and applied to pump-turbine applications, both in pump and turbine mode.

The major advantage of this approach is its robustness, including the ability to handle back flow at the interface, and accuracy. Compared to currently available methods, both in proprietary and commercial codes, the implicit approach leads to a consistent treatment of the interface, enforcing natively the idea of the mixing-plane, i.e. constant spanwise-distribution of the quantities. This allows to obtain excellent results also at part- and over-load.

1. Introduction

In the design process engineers look for fast tools able to produce a reliable picture of the fluid dynamics of the components. Here not only the design point is of interest, but also off-design conditions, which in the last decades have become more and more important and should be assessed in the preliminary phase of the process. At these operating points the flow can be quite different compared to design conditions, presenting considerable non-uniform distributions of pressure and velocity. Even backflow is often encountered at the interface between two components so that the involved CFD method needs to be able to produce accurate results at these, numerically-speaking, unfavorable conditions.

The usual approach in order to be fast in the design procedure is a steady-state computation, mainly using a mixing plane approach based on the reference papers of Denton [1, 2, 3, 4]. As soon as backflow at the interface is present, the usual method based on the definition of additional (explicit) boundary conditions at the mixing-plane interface leads to numerical problems, since on the same boundary the flow can have two directions, thus requiring considerable effort from the numerical point of view in order to obtain a physically sound solution.

In the present paper a fully implicit approach for the mixing-plane interface is presented and



applied to hydro-machines examples. Being fully implicit, the proposed method do not base on the definition of additional boundaries, which results in an increased numerical stability of the interface. Additionally, the above mentioned problems encountered with classical methods in case of backflow are directly avoided through the new approach.

2. Numerical discretization

The solver used is an in-house 3D, unstructured, finite volume code based on the object-oriented open source toolbox OpenFOAM[®]. The solver is a segregated, steady-state SIMPLE-C incompressible solver using a multiple reference-frame (MRF) formulation. The choice of the SIMPLEC pressure-velocity coupling presents a considerable improvement in robustness and accuracy compared to the standard SIMPLE technique or the projection method used originally in OpenFOAM[®] V2.2 for incompressible flows.

3. Mixing Plane Theory

In order to reduce the computational costs of turbomachinery calculations one of the most often used simplifications is the use of rotational periodicity, thus computing only a single-passage. Since most often the periodicity of the rotor differs from the stator the exchange of flow variables at the interface needs special treatment.

One of the most common is the mixing plane approach. In turbomachinery applications the flow is largely mixed out in the gap between two consecutive blade rows. This fact thus is used in the mixing-plane approach by imposing pitchwise uniform total conditions at the interface in order to mimic this behavior. The variables are therefore circumferentially averaged before exchanging the information with the other interface side.

3.1. Explicit Implementation

Today's implementations of mixing planes are almost always based on an explicit or semi-explicit treatment of the interface boundaries. The interfaces are treated as additional in- and outlets leading to a numerical separation of a physically connected system. For simple rotor-stator configurations the systems of linear equations would read as equation 1:

$$\underline{\underline{A}}_{rotor} \cdot \underline{x}_{rotor} = \underline{b}_{rotor} \quad \text{and} \quad \underline{\underline{A}}_{stator} \cdot \underline{x}_{stator} = \underline{b}_{stator} \quad (1)$$

The coupling at the mixing plane is treated as completely explicit inside of the source term \underline{b} . Explicit contributions to linear system of equations are kept constant during inner iterations and are only updated during outer loops. Especially at the beginning of an iterative numerical procedure this sudden change in the source term can lead to instabilities, often leading to divergence. Additionally, a correct evaluation of the type of boundary conditions for each variable as well as its value can be difficult.

The averaging at the interface has to account for conservativity constraints of the NS-equations and the type of boundary condition has to allow backflow through the interface as well as local changes in the characteristics based on the flow speed.

The instabilities are also often referred to be a problem of a missing appropriate modeling of the far field boundaries, leading to unphysical incoming disturbances. The solution to this problem is generally known under the name "non-reflective boundary conditions" (NRBC) and many authors have investigated in this field, e.g. [5, 6, 7].

The present authors indeed consider this attempt as misleading since it is a users choice to

impose uniform total conditions and actually the very basic concept of the mixing plane. Superposition of local variations lead to local deviations of the total pressure, see [8], even if in the mean constant total pressure is preserved. Although such a treatment may improve stability of the numerical procedure it weakens the mixing plane constraints which are a key feature for correct steady-state predictions of turbomachinery load.

3.2. Implicit Implementation

All these drawbacks of the common implementation of the mixing plane motivated the present authors to investigate on a new implementation. It is based on a fully implicit implementation, where the influence of the mixing plane is accounted by adding the interface coupling coefficients A_C to the matrix of internal coefficients A_I . Numerical domains connected by mixing planes can therefore be integrated into one system of equations making the treatment of cells at the mixing plane equal to any other internal cell of the computational domain.

$$(\underline{A}_I + \underline{A}_C) \cdot \underline{x} = \underline{b} \quad (2)$$

Figure 1 shows a simplified rotor-stator configuration for a better illustration of this process. Coefficients for the coupling matrix A_C , e.g. on the rotor side, are built based on local variables a_i and average variables of a ghost-cell \bar{b} corresponding to the pitchwise average of the stator.

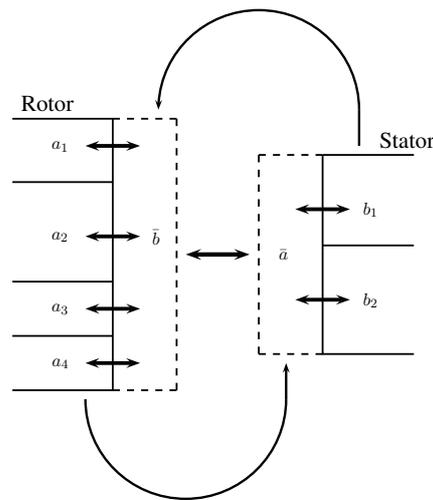


Figure 1: DISCRETISATION OF INTERFACE FLUXES

By this change in the numerical treatment the above stated problems and their various solution attempts could be solved at once. No additional numerical treatment is needed to avoid numerical artifacts, enforcing a strong numerical coupling of the components. It allows to have a fully consistent system based on the accuracy and the physics of the equations solved in the domain. The resulting implicitly connected system is therefore able to satisfy both at the same time, the flow physics and the mixing plane constraint.

There is no additional user input needed on the orientation of the interface or whether its mapping considers passages or complete 360° interfaces. For more details regarding the implementation and additional profound validation of the numerics please refer to previous publications [9, 10] of the authors.

4. Case description

4.1. Numerics

All presented cases are computed in steady-state with the proposed mixing plane interface. A high resolution, second order scheme has been used, together with the SST turbulence model using automatic wall treatment as well as a k- ϵ model using wall functions. In pump mode one runner (RN) passage (of 9) and one guide-vane (GV) passage (of 20) are computed (see Figure 2 (a)), with a mixing-plane interface between them, while in turbine mode also the flow in the draft tube is considered (see Figure 2 (b)), using a runner passage vs. full-circumference interface (i.e. 40 vs. 360 degrees, see Fig. 3 for details). The mesh in pump mode exhibits a pinch in the GV, in order to reduce the possible numerical problems at low flow rate when backflow is expected at the computational-domain outlet.

The mesh information including y^+ is given in Table 1. These are approximate values which vary slightly with varying stagger angle of the guide vanes.

	Runner	Draft tube	Wicket Gate
# cells	$6 \cdot 10^5$	$8.5 \cdot 10^5$	$2 \cdot 10^5$
y^+ values	50 (walls) < 20 (blade)	< 100 (walls)	< 150 (walls) < 20 (blade)

Table 1: Mesh information

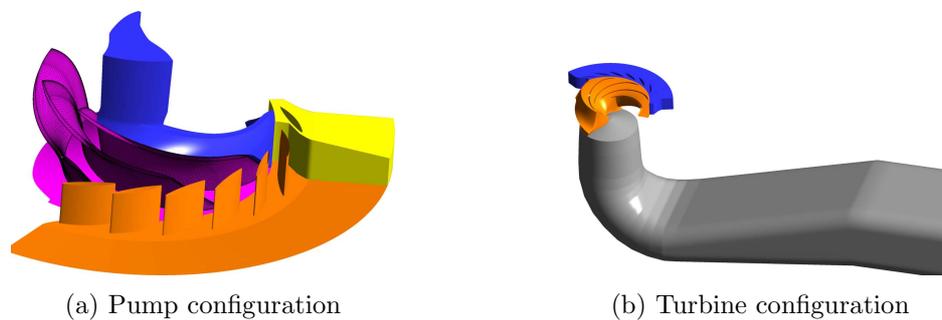


Figure 2: Computational Domains

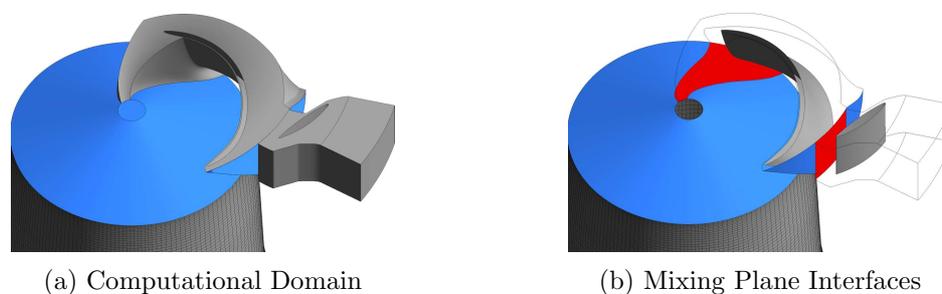


Figure 3: Interfaces. Example for turbine case. Red: upstream interface; Blue: downstream interface.

4.2. Turbine-Mode

In turbine-mode six different guide vane positions were simulated, imposing the flow rate according to measurements and corresponding to a constant $\psi=0.94$.

This configuration includes two mixing planes, ones in radial direction from the guide vane to the runner passage and ones as a one-passage to 360° axial interface from the runner to the draft tube.

Global results The computed results are presented as pressure-coefficient ψ and normalized efficiency η_{norm} for the different GV openings in Fig. 4 corresponding to different flow coefficient values. The pressure coefficient shows a nearly constant behavior, within less than 2% of the expected (i.e. measured) constant average. The results with the two turbulence models are comparable, with the SST-model showing slightly smaller differences to the measurements.

Considering the efficiency the SST model clearly presents a superior behaviour, capturing very well the overall shape, while the $k-\epsilon$ model is too optimistic at medium flow coefficient while producing too high losses at low flow conditions.

In Fig. 4 results obtained with a commercial solver are also presented, for both pressure coefficient and normalized efficiency. As it can be seen, the difference to the measured data is for both CFD codes in the same range, thus underlining the quality of the proposed method.

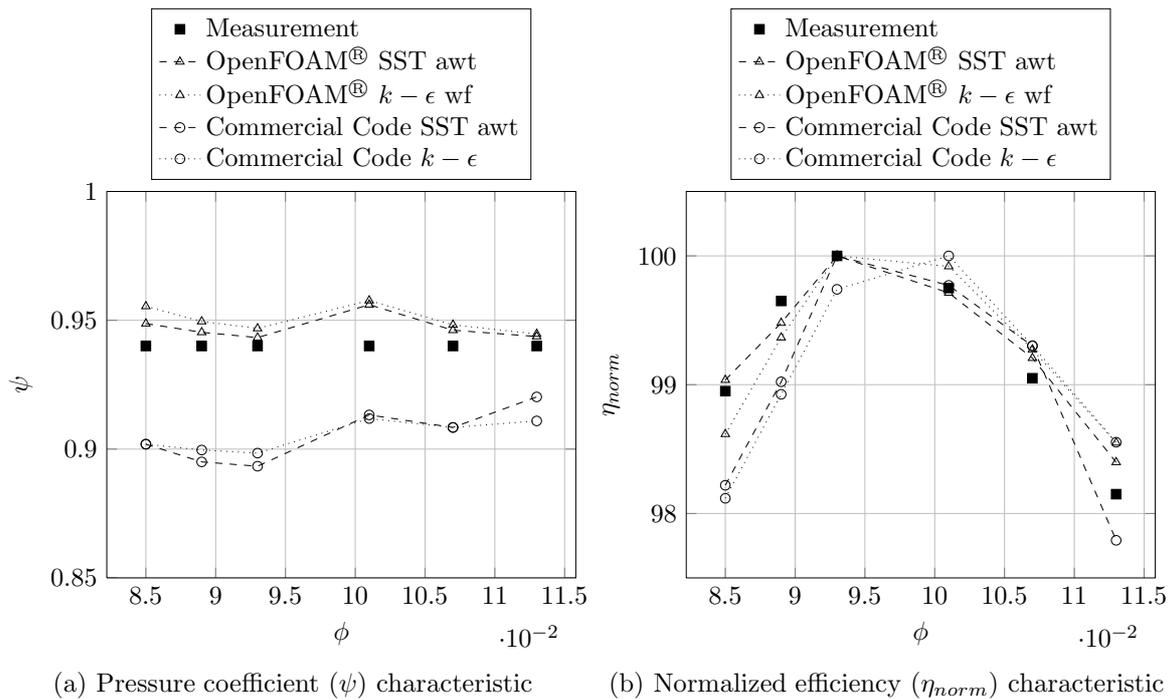


Figure 4: Pressure coefficient and normalized efficiency

Local results Looking at the local flow behaviour, Fig. 5 shows the low pressure region in the draft tube at low and high flow coefficient. While at high flow rate the low pressure region is well centered indicating axisymmetric conditions, at low ϕ there is a clear sign of asymmetry as a precursor for the unsteady rotating vortex rope.

Streamlines started in low-energy flow regions behind the elbow are also presented. At minimum ϕ a backflow zone could be detected, indicating a clearly non-uniform flow in the draft-tube, while at high flow rate a nearly symmetric vortex pair and corresponding total pressure distribution were found, indicating a stable, well-behaving flow behind the elbow.

Considering a blow-up of the interface between runner and draft-tube in Fig. 6 for the smallest GV-opening (i.e. low ϕ), a backflow zone is detected. For this operating point convergence was below average due to the fact that a vortex rope is starting to develop, thus driving the computation to an unsteady state and reducing the steady-state convergence because of local effects. Nevertheless, considering the efficiency in Fig. 4 (b) the global behavior is still very well captured with the steady-state approach. Fig. 6 (b) presents the total pressure distribution at the 360 degree interface on the draft tube and on the interface between runner and GV, on the runner side. The circumferential uniformity of the solution (up to the mesh resolution) can be clearly recognized, showing that the proposed interface is able to impose a constant circumferential distribution thus fully satisfying the original condition behind the mixing-plane approach. Comparing the results with codes using the classical explicit approach it can be clearly stated that being able to satisfy this condition is crucial for good results, especially at off-design conditions.

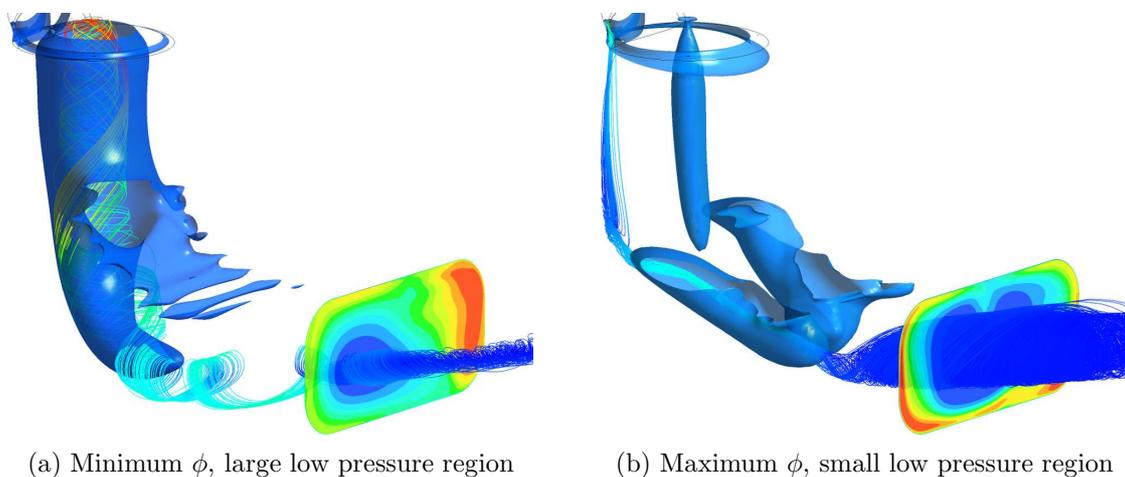


Figure 5: Identification of low pressure regions (blue) in the draft tube with pressure isosurface

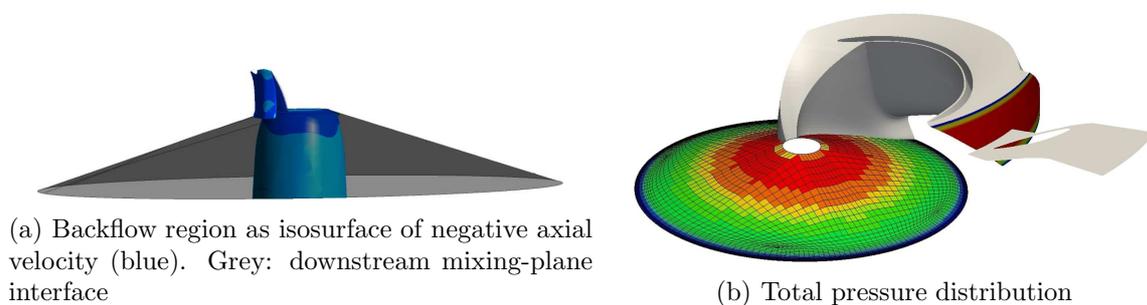


Figure 6: Flow features at the mixing planes

4.3. Pump-Mode

In pump-mode a speed-line has been computed, keeping the GV position fixed at 20 degrees and imposing the flow rate at the inlet for five operating points.

Global results The pump characteristic (speed line) is depicted in Fig. 7 as $\psi(\phi)$, and compared to measured data. Since the measured data includes also the spiral casing, a difference between measured and computed results is expected due to the negative slope of the spiral-casing characteristic. This fact is emphasized especially at high flow rates.

Also in pump mode two speed-lines were computed using both SST and $k-\epsilon$ turbulence models. The curves are practically parallel and show the same general behaviour. Nevertheless there is a difference in the level of the results, with those computed with the SST model showing clearly higher values. This fact should be attributed to the higher dissipation produced by the $k-\epsilon$ compared to the SST model.

In order to compare the computed results with the measured one, the characteristic of the spiral casing was extrapolated fitting the difference between measured and computed data with a second-order polynomial, which should approximate the losses in the spiral casing. The so corrected characteristic is also presented in Fig. 7 and shows for four of the five operating points a very good agreement with the measured data, indicating the general validity of the assumption and the quality of the computed results. At the highest flow rate there is a discrepancy between CFD and experiments, which is attributed to additional losses generated at the spiral-tongue at high incidence. These are of course not included in the simple approximation of a constant loss-coefficient assumed for the correction of the numerical results.

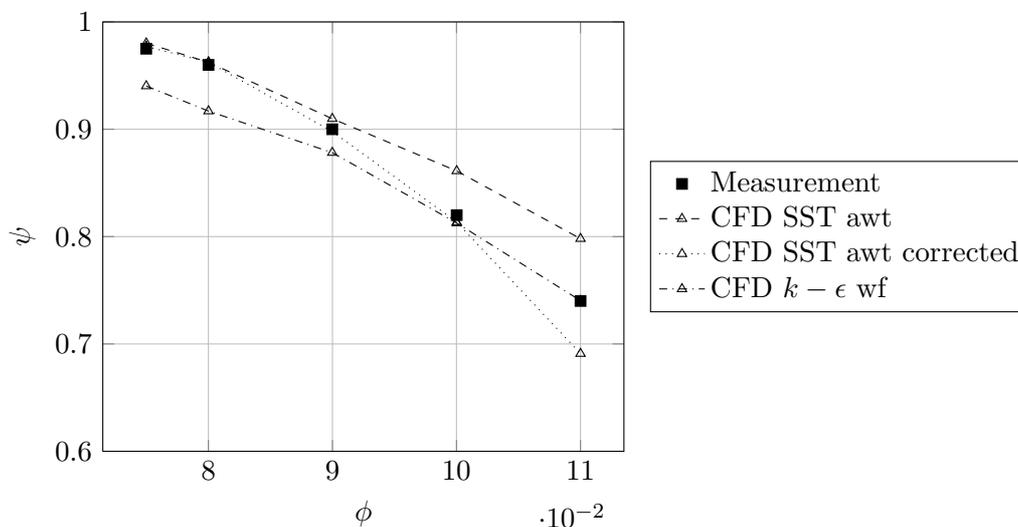


Figure 7: Pressure (ψ) and flow (ϕ) coefficient characteristics

Local results At low flow-rate (last two operating points) a backflow zone at the exit of the computational domain is detected and handled numerically with an opening boundary-condition, imposing in case of outflow a constant pressure, and in case of inflow a total-pressure and normal direction.

The backflow is already present in the vaneless space between runner and GV, as can be seen in Fig. 8. Please note that the backflow zones on the interface (i.e. on runner and GV side) are different due to the local effect of the potential field of the GV respectively runner and the

imposed constant total pressure at the mixing-plane. This allows the flow to adjust locally while maintaining the mixing-plane constraint, i.e. constant circumferential average.

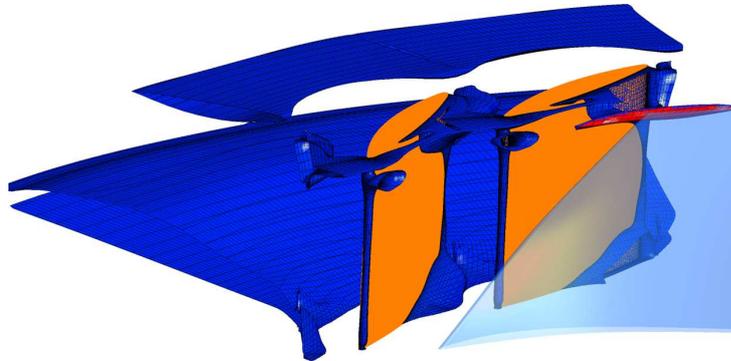


Figure 8: Backflow regions in GV (pump mode) as isosurface of negative radial velocity (blue)

5. Conclusions

In the present paper a novel mixing plane has been presented and applied to the flow in reversible pump-turbines, both in pump and turbine mode.

The novel approach is based on a fully implicit handling of the interface, thus leading to increased speed and robustness of the computations. Moreover the proposed interface is able to handle backflow natively, i.e. without particular care as it has to be used with usual explicit approaches, which are based on the definition of additional boundary conditions at the interface. The validation of the results against measured data shows the validity of the approach for a wide range of operating conditions, thus underlining his strength as effective tool in the design process, where fast but also reliable results are needed.

References

- [1] Denton J 1985 *AGARD lecture series, Vol.140*
- [2] Denton J 1992 *Journal of Turbomachinery, January 1992, Vol. 114/19*
- [3] Denton J 2010 *Proceedings of ASME Turbo Expo 2010*
- [4] Denton J and Singh U 1979 Time marching methods for turbomachinery flow calculation *In Von Karman Inst. for Fluid Dyn. Appl. of Numerical Methods to Flow Calculations in Turbomachines 47 p (SEE N80-12365 03-34)* vol 1
- [5] Engquist B and Majda A 1977 *Proceedings of the National Academy of Sciences* **74** 1765–1766
- [6] Giles M 1988 *TR-88-1*
- [7] Holmes D 2008 *ASME paper GT-2008-51296*
- [8] Saxer A and Giles M 1991 Quasi-3-d non-reflecting boundary conditions for euler equations calculations *AIAA Computational Fluid Dynamics Conference, 10 th, Honolulu, HI* pp 845–857
- [9] Hanimann L, Mangani L and Casartelli E 2014 *Journal of turbomachinery* **136** 400–407
- [10] Hanimann L, Mangani L and Casartelli E 2014 Reviewing the implicit mixing plane approach *ASME Turbo Expo 2014: Power for Land, Sea and Air* American Society of Mechanical Engineers (ASME Paper GT2014-25869)