Proceedings of the 11th Brazilian Congress of Thermal Sciences and Engineering -- ENCIT 2006
Braz. Soc. of Mechanical Sciences and Engineering -- ABCM, Curitiba, Brazil,- Dec. 5-8, 2006

Paper CIT06-0450

PHYSICAL AND NUMERICAL ANALYSIS OF NAVIER-STOKES ROTOR-STATOR INTERECTION MODELS

Alex GUEDES
Instituto de Estudos Superiores da Amazônia. Quadra 32, No. 10, 66640-255. Mangueirão, Belém-PA, Brazil.

Jean-Louis KUENY
Laboratory for Hydraulic Machines, Swiss Institute of Technology. Av. de Cours 1007, Lausanne, Switzerland.

Abstract. The parallel rotor-stator computation capabilities of Fine Turbo and CFX-TASCflow CFD commercial codes have been investigated within the frame of the European ESPRIT project HPNURSA. Unsteady computations have been performed for simple test cases, where flow conditions are known, the numerical results indicate that the unsteady rotor-stator interaction model is capable of accurately reproduce the features of the incompressible flow encountered in hydraulic turbomachines. It is also shown that R-S computations are very demanding in mesh refinement if small scales flow structures must cross the R-S interface. Finally, the behaviour of an industrial pump-turbine, by VA Tech Hydro, is investigated using CFX-TASCflow. The unsteady computation results were validated by LDV and PIV measurements, showing that unsteady R-S computations could reproduce the experimental flow features for nominal and off-design operation conditions for this pump-turbine.

Keywords. Pump-turbine, Rotor-stator, Navier-Stokes, Parallel computation, Numerical method.

1 Introduction

Accurate numerical prediction of the flow field of a pump-turbine can allow the design of more compact and silent turbomachine, with an increased operation range, specially for pumping mode, where part load instability are common.

Many authors indicate that rotor stator interaction is paramount for pumping systems at off-design conditions, e.g. Eisele K et al. (1998), Kaupert (1996 and 1997). Recently, different authors analysed the unsteady rotor-stator interaction. For instance, an experimental and numerical analysis of the runner-diffuser interaction in a centrifugal pump has been performed using LDV and PIV by Ciocan et al. (1998 and 2000) and Dong et al. (1997). Other experimental analysis of flow between impeller and bladed diffuser using LDV technique or pressure transducer are presented in Toussaint et al (1998) and Ubaldi et al. (1996).

All these works show the unsteady nature of the rotor stator flow in pump systems, therefore, it is of prime importance to investigate the capability of the rotor-stator interaction models recently implemented in leading CFD commercial codes to represent unsteady phenomena. It is also important to study the numerical characteristics of these codes and its suitability for designing industrial turbomachines.

The following sections show an investigation of rotor-stator computation capabilities of Fine Turbo and CFX-TASCflow CFD commercial codes, developed respectively by NUMECA international and AEA Technology. This is done by two test cases, where flow conditions are known. Then, CFX-TASCflow code is applied for analysing the nominal and off-design behaviour of an industrial pump-turbine designed by VA Tech Hydro. The results for the pump-turbine flow computations are presented and validated by experimental measurements by PIV and LDV techniques.

All the numerical results shown in this work were obtained in the frame of the European ESPRIT project HPNURSA - “High Performance Numerical and Unsteady Rotor Stator Analysis”.

2. Rotor Stator Interaction Models

Both codes use a sliding surface approach for modelling the rotor-stator interaction, where all interaction occurs over a revolution surface that separates the rotating and non-rotating domain, see Fig. (1). There are three models that differ in the way flow field information is passed across the sliding surface. The first one is the mixing plane model (referenced here as mean stage model), where flow field variables are integrated over tangential direction, so only steady interaction is possible. The second one is the quasi-steady model (referenced here as frozen rotor model), where the appropriated frame changes for vector variables is performed, but rotor-stator relative position is not updated at each time-step. Finely, the unsteady model that accounts for frame and relative position changes and is the only that can be expected to fully reproduce the features of the real rotor-stator flow.

More details of the implementation of these models for CFX-TASCflow can be found in Kuntz (2000).

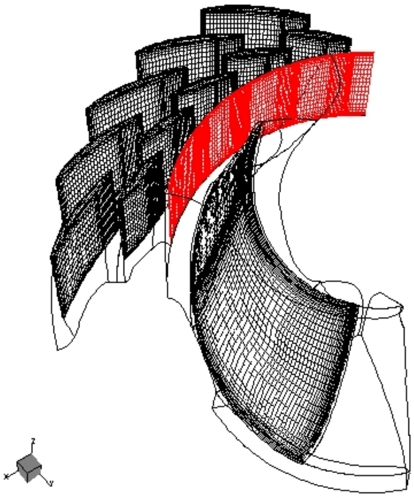


Figure 1. Sliding surface for rotor-stator computations of VA Tech pump-turbine.

3. Rotor-stator models validation - Steady perturbation Test Case

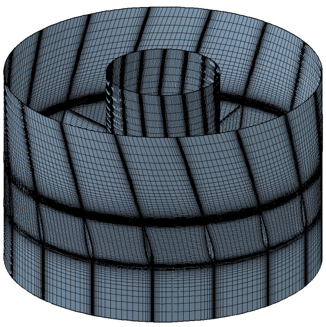
The main objectives of the academic test cases is to control the quality of the Rotor-Stator interfaces implemented in the different CFD codes and to define its numerical limits to ensure a correct transfer of physical quantities through the interface for simplified geometries and flows, but still representative of typical flow in turbomachines.

Two steady state test cases have been defined. For these geometries, the perturbation of a uniform flow field is only generated on one side of the interface and gives rise to a steady perturbation in the corresponding static or rotating frame. This steady flow perturbation must be unchanged after and before the interface. We analysed a blade wake crossing the rotor-stator interface and also the upstream pressure perturbation before a blade. A typical axial and radial *turbo machines* geometries have been defined to analyse the numerical flow behaviour in these two classical turbomachine configurations, as found in Henry P. (1992). For these two test cases the numerical unsteady rotor-stator flow will be compared to the steady flow obtained by non rotor-stator computation, with the same geometry and mesh. As the computations for axial and radial test cases present qualitatively the same behaviour and lead to the same conclusions, only results for the axial configuration will be presented.

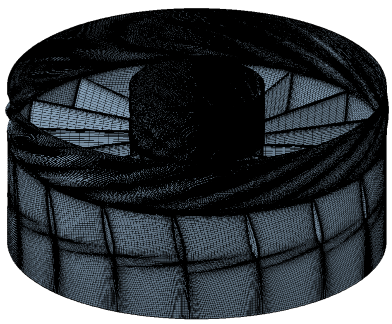
Six different meshes have been built using ICEM mesh generator or IGG+AUTOGRID, as summarised in Table (1), for ensuring the quality of the numerical results. Some typical meshes are presented on Fig. (2).

Table 1 Meshes for axial rotor-stator test case

Mesh Name	Number of Nodes
RS-1	131175
Axial-Fix (direct or flow aligned)	112791
BIG (flow aligned)	221375
BIG2 (flow aligned)	288015
BIG3 (flow aligned)	238098



A) Axial-directed.



B) Flow aligned mesh

Figure 2. Meshes for steady state test case.

The computations have been performed on a WINDOWS NT computer (Pentium III, 1GHz, 512 M Ram). For a mesh size of about 240'000 nodes, the CPU time per iteration is 0.95 ms, then 2 to 3 CPU days are necessary to obtain convergence if 5 iterations per time step are performed.

Typical results for rotor-stator computations showed the desired continuity and conservation properties of the rotor-stator interface models. But it has been shown that continuity is strongly dependant of mesh refinement in both sides of the R-S interface. For large-scale flow structures, like the one seen in Fig. (3), as for potential effects shown in Fig. (4), the continuity is assured for every rotor stator relative position. For small scale structures, the continuity is only assured for rotor-stator relative positions where mesh is refined enough for capturing the flow structure in both side of the R-S interface, as seen in Fig. (5a), where the refined zones are aligned and the wake starts to cross the R-S interface, on the other hand, for another time step, shown in Fig. (5b), the refined zones are non aligned and the trailing edge wake cannot traverse the R-S interface.

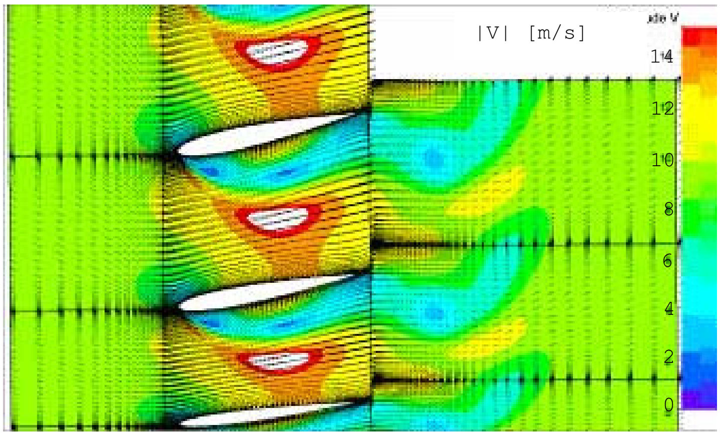


Figure 3. Large vortex crossing the R-S interfaces.(Fine Turbo computation)

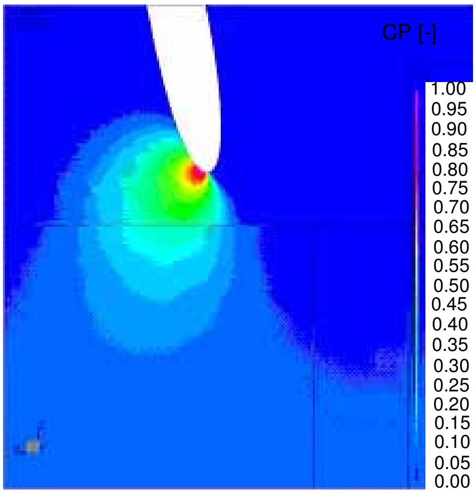


Figure 4. Upstream pressure perturbation crossing the R-S interface. (Fine Turbo computation)

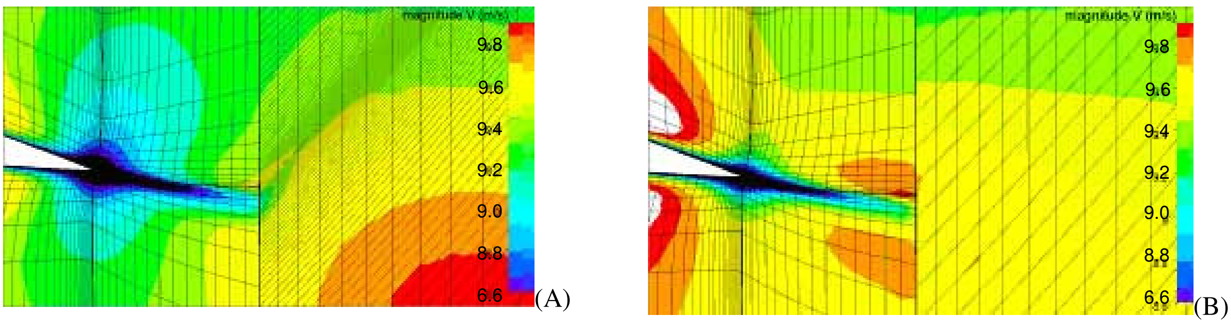


Figure 5. Velocity magnitude field at 2 time steps. (Fine Turbo computation)
(A) The wake crosses R-S through the refined mesh in both sides. (B) The wake is diffused by coarse mesh.

It is important to note that the limitation with continuity across the R-S interface for small scale flow structures is a pure mesh refinement effect and is not related to the R-S interaction models implemented in the commercial codes, as this same limitation can be found in non R-S computations as well, but for unsteady computations, the relative R-S position changes at each time step, then the only way to assure good spatial discretization is to have a fine mesh all over the angular extension of the rotating domain. This fact makes very resource intensive when one is interested to catch small-scale structures, because good refinement should be achieved for every R-S relative position. One possible solution is to apply mesh-adapting techniques as the relative position changes.

4. Rotor-stator models validation - Unsteady perturbation test case

An unsteady test case in a rotating frame has also been defined. The unsteadiness corresponds to periodic vortex shedding generated by a square section body in an uniform axial flow field, as can be seen in Fig. (6), with flow parameters based on a classical vortex shedding setup, for witch experiments and multiple CFD solution are available. The simulation is done in 60 deg. channel periodicity; the square side is 0.1m, the Reynolds number is 22000 and the Strouhal number is 0.132. This test case allows verifying possible influence of the rotor-stator interface on unsteady behaviour of the flow, in particular the vortex shedding frequency and the vortices advection.

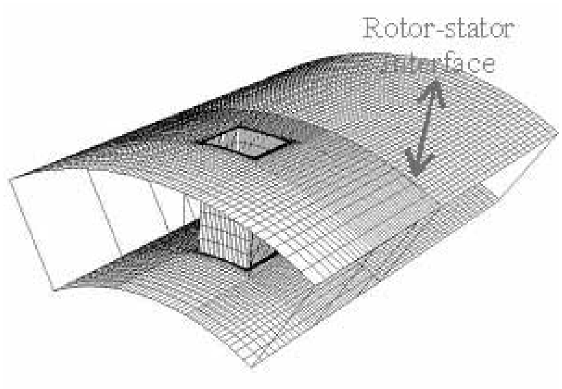


Figure 6. Mesh for the unsteady test case.

The goal of this test is not to reproduce the experimental unsteady features of the flow, as it does not depends on rotor-stator interaction model, but only investigate if the computed unsteady flow field changes as a result of the introduction of the R-S interface. Fig. (7) and Fig. (8) show respectively the instantaneous velocity field for a computation with and without rotor-stator interface, comparing both cases, one can see that the shape and continuity of the vortex structures shaded past the square body is achieved by R-S computation. The shading frequency was exactly the same in both cases as it is driven by the flow conditions around the square body; witch is located entirely in the stator domain and is not influenced by the introduction of the R-S interface.

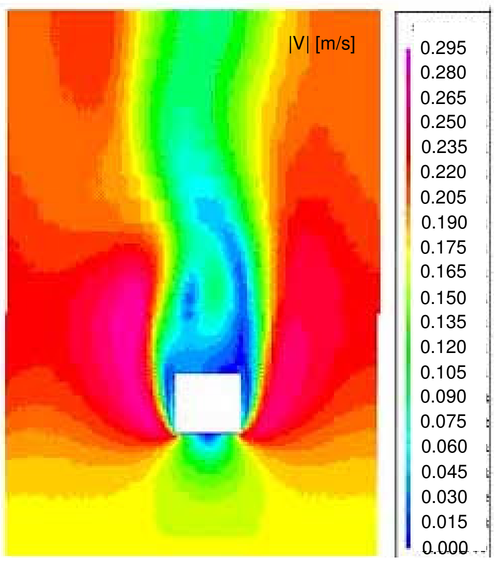


Figure 7. Flow field for unsteady non R-S computation. (CFX-TASCflow computation).

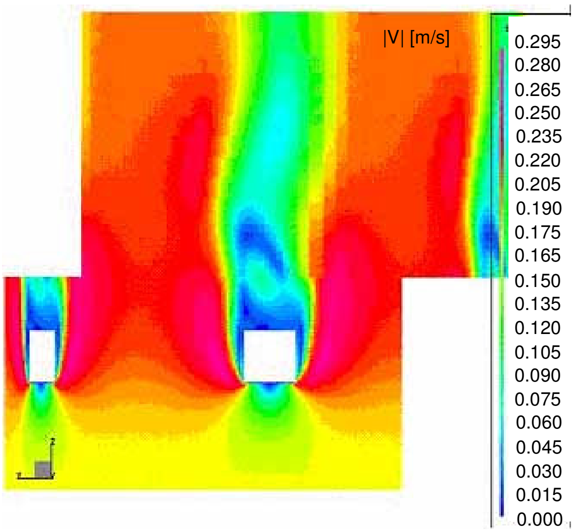


Figure 8. Flow field for unsteady R-S computation. (CFX-TASCflow computation).

The results obtained for steady and unsteady test cases by both Fine Turbo and CFX-TASCflow show that the rotor-stator interaction models implemented in these codes are equivalent to the limits of these tests. Furthermore, considering the results of both test cases together, it was shown that the rotor-stator models implemented are fully conservative and capable of reproducing the typical flow conditions encountered in the rotor-stator interaction zone of a hydraulic turbomachine.

5. Pump-turbine analysis

The Francis type pump-turbine studied has specific speed Nq=66, with 5 runner blades and 22 guide-vanes, as can be seen in Fig. (9). The computations for the pump-turbine were all performed for pumping mode operation, for which this turbomachine displays strong instability behaviour at part-load operation at a guide-vane opening of 18deg., as shown in Fig. (10).

Figure 9. Pump-Turbine geometry.

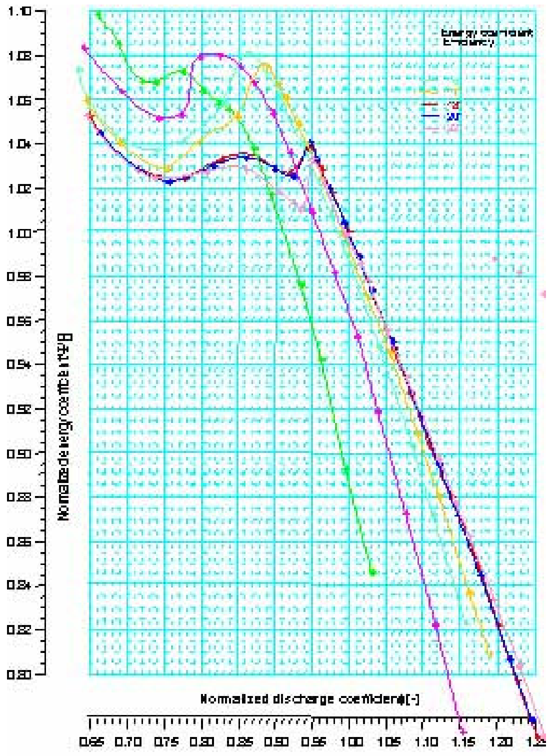


Figure 10. Pumping characteristic curves.

The aim of this study is to show that rotor-stator interaction models implemented in the commercial CFD codes can predict all the complex 3D unsteady turbulent flow features encountered in off-design operation of an industrial pump-turbine, allowing a better understanding of the behaviour of such a turbomachine. To achieve this goal, a numerical simulation campaign, as well as experimental flow field measurements with Particle Image Velocimetry and Laser Doppler Velocimetry techniques were performed over four different operating points, comprising full discharge, nominal and part-load operation.

6. Numerical simulations

For computational resources limitation reasons, all the computations for the pump-turbine were performed using CFX-TASCflow CFD software, which is a structured finite volume solver for the incompressible Reynolds averaged Navier-Stokes model, along with the standard k-□ model for turbulence closure.

The original geometry has not exact periodicity, as we have 5 runner channels and 22 stator channels. A different stator configuration was used for unsteady computation, corresponding to a 22 stator channels, allowing the use of periodicity condition for 1 runner channel and 4 stator channels, reducing greatly the computational resources needed. The final domain configuration for unsteady computations can be seen in Fig. (11).

The boundary conditions for the computations are of steady state type, as summarised in Table (2), so the unsteadiness is generated only by rotor-stator interaction.

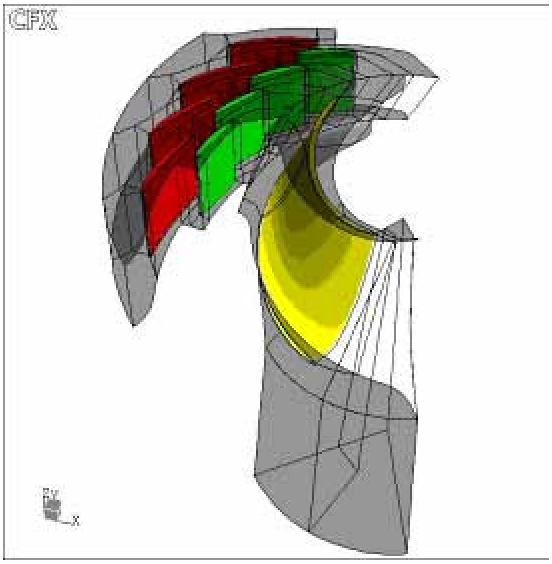


Figure 11. Pump- turbine computation domain.

Four meshes refinements were tested for ensure the non-sensitivity to meshes effects, their names and respective number of nodes is shown in Table (3).

Table 2 Boundary conditions for the Navier-Stokes computations.

Boundary condition	**Information supplied**
Inlet	Flow rate and direction
	Turbulence intensity and length scale
Outlet	Average static pressure
Solid walls	Non slip / Log law.

Table 3 Meshes configuration.

R-S Model	**Grid Configuration**	**Number of Nodes**
Mean Stage	1 runner channel + 1 stator channel	230 886
Frozen Rotor	1 runner channel + 5 stator channels	356 901
Unsteady	1 runner channel + 4 stator channels	1 019 364

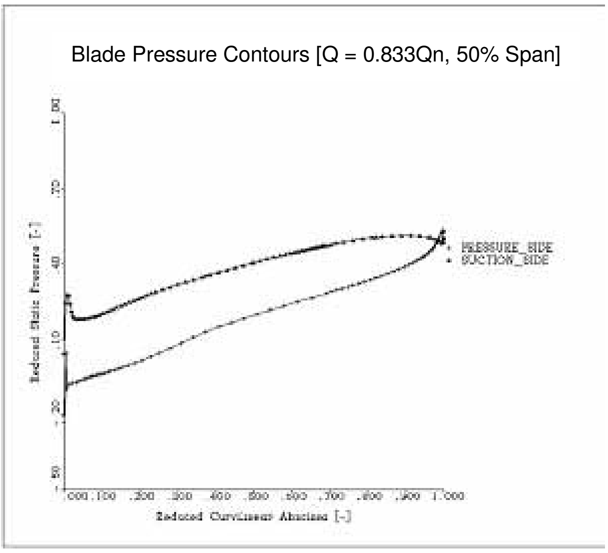
For steady state applications, a suitable time step size is $0.5/\Omega$, where Ω is the rotational frequency of the computation. For transient simulation, the time scale of the transient effects has to be estimated. Alternatively, it can be expressed by the number of time steps necessary to cover one runner blade passage, which should be not bigger than the number of grid cells in tangential direction over the rotor-stator sliding surface, for achieving a good relationship between spatial and time resolution. This results in the following time step expression:

$$\Delta t = \frac{2\pi}{\Omega}\frac{1}{N_{blades}N_{step}},$$

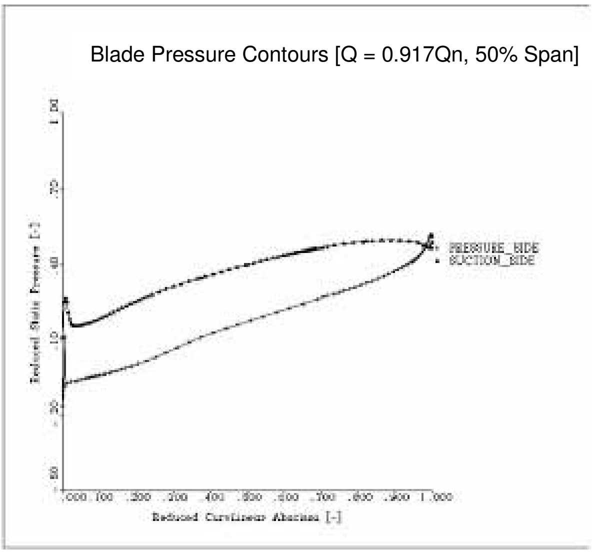
where N_{blades} is the number of blades of the machine and N_{step} the number of time steps per blade passage

7. Flow analysis

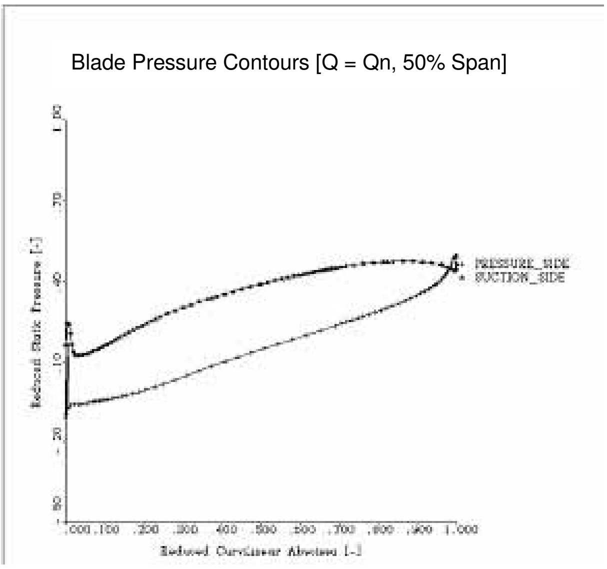
First it is presented some results for the different operating points in order to assert the flow field change as the flow rate – Q - reduces. The Fig. (12) shows the contours of the pressure coefficient over the blade of the runner four operating points, at the middle of the channel in spanwise (hub to shroud) direction. One can see increasing chocks effects on the leading edge as the flow rate reduces.



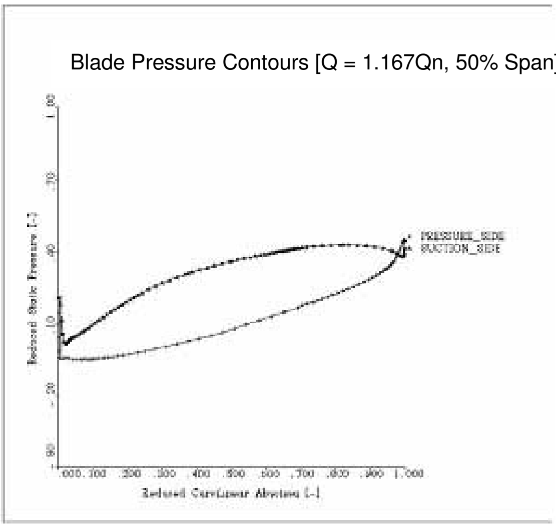
A) Q=0.83Qn.



B) Q=0.9Qn



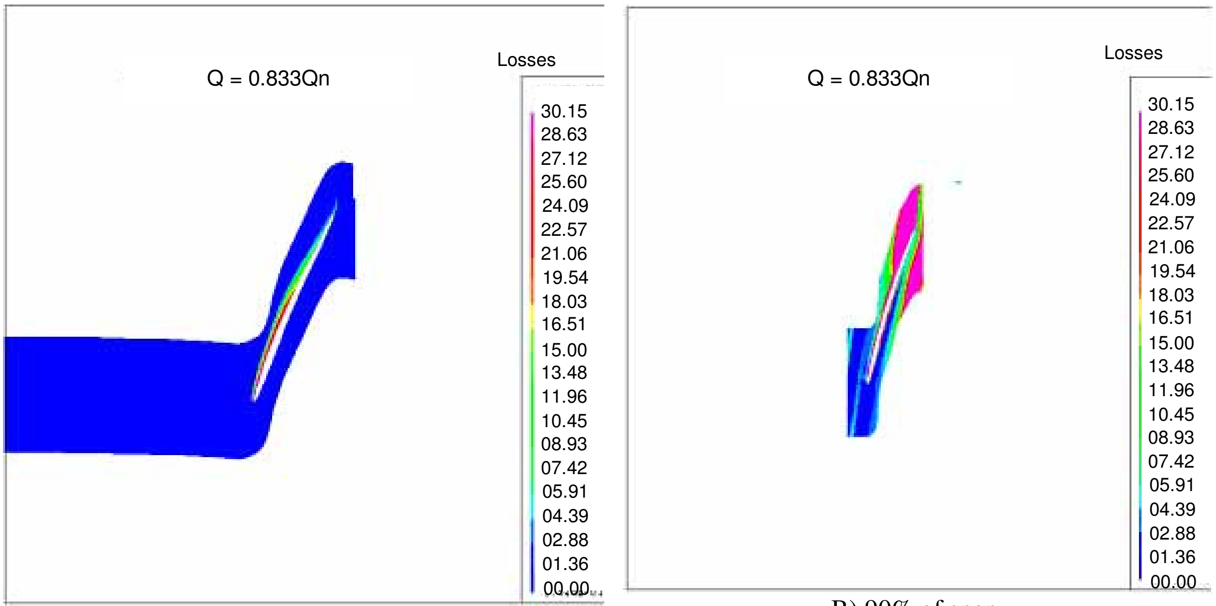
C) Q=Qn.



D) Q=1.17Qn.

Figure 12. Pressure coefficient at 50% of span.

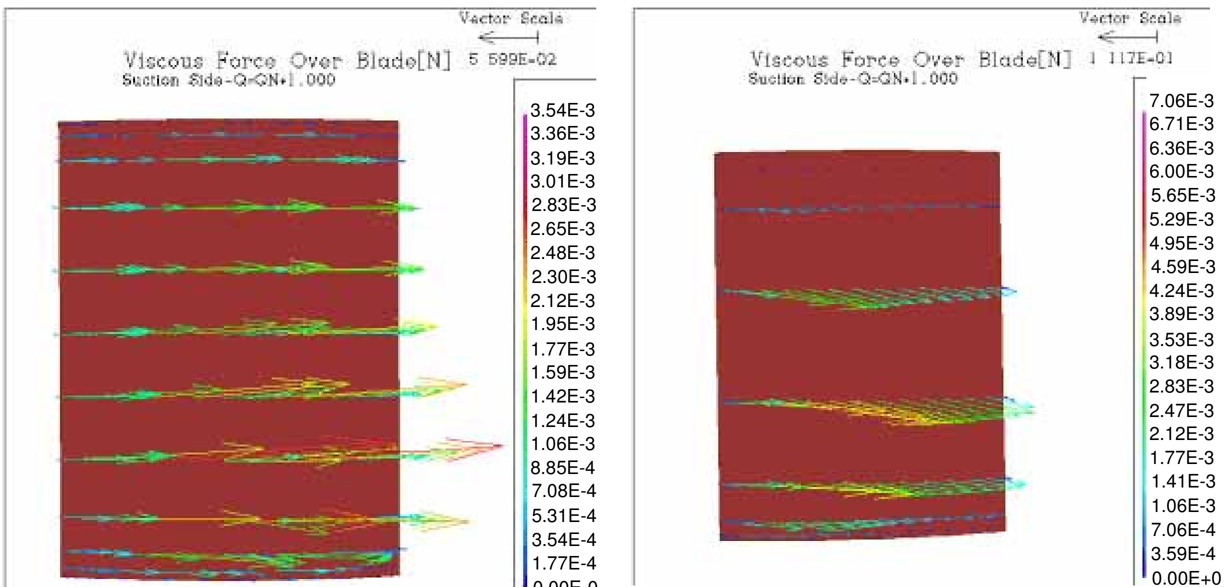
On Fig. (13) it can be seen that the hydraulic losses over blade to blade surfaces in the runner is concentrated in the shroud side for the operating point Q=0.83Qn. This leads to non-adapted flow angles entering the stator domain near the shroud surface.



A) 10% of span. B) 90% of span.

Figure 13. Hydraulic losses for Q=0.83Qn.

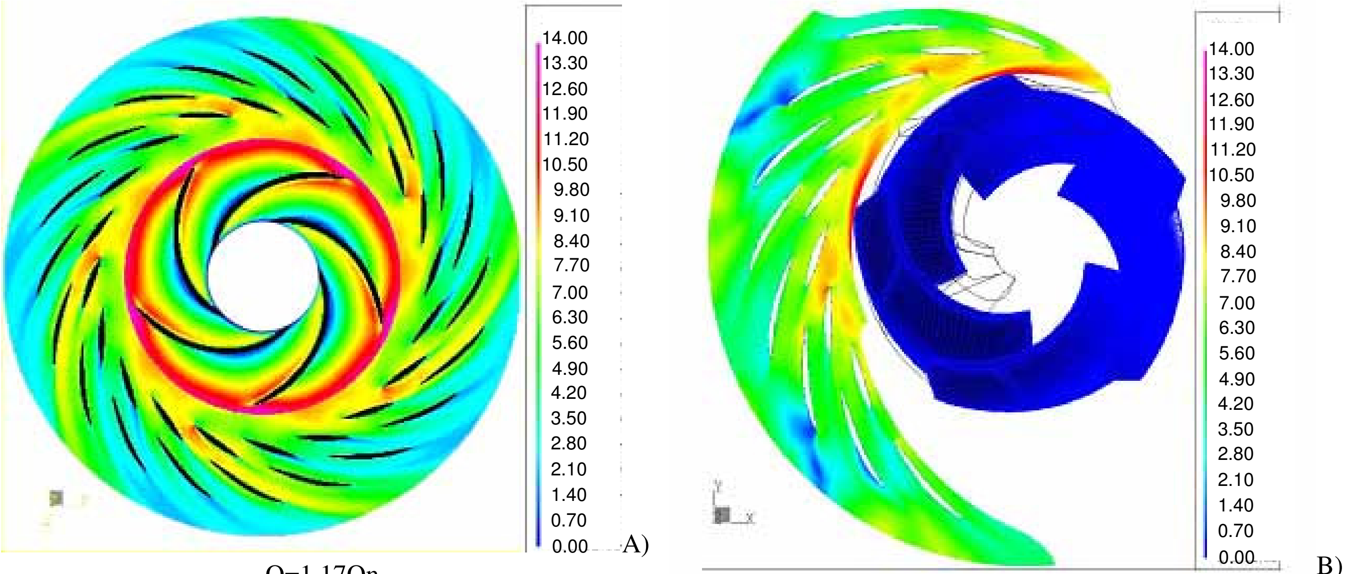
Another interesting result is seen in Fig. (14). The prediction of the viscous force over the guide vane is strongly influenced by the rotor stator interaction model, as for the mean stage model (Fig. 14A), the direction of the flow entering the stator domain is given by its □-integrated value over the rotor-stator sliding surface. This mean flow orientation can greatly differs form the instantaneous flow for a given rotor-stator relative position (Fig. 14B) even for the nominal operating point.



A) Mean stage model. B) Unsteady model.

Figure 14. Viscous force over the suction side of the guide blade, coloured by |V| [m/s].

The instantaneous absolute speed colour map displayed on Fig. (15) shows that the flow field in the stator zone is highly non-uniform, with noticeable changes from one channel to another. For both operating points Q=1.17Qn (Fig. 15A) and Q=Qn (Fig. 15B) it can be seen that, for a given R-S relative position, the flow around the leading edge of the guide vanes has good or bad incidence according to the channel considered; it can be seen as well that there is a low speed zone close to the trailing edge of the stay-vanes, show a tendency of flow detachment. These flow non-uniformities are more important for off-design operation conditions (Fig. 15A) when compared to the nominal operation point (Fig. 15B).



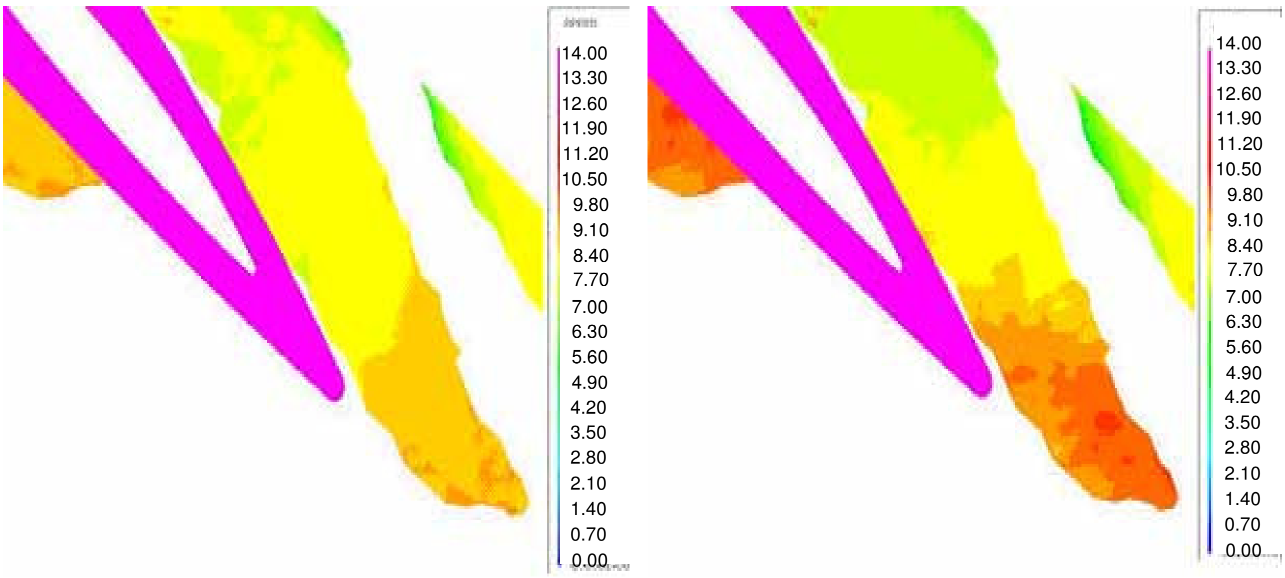
Q=1.17Qn.   Q=Qn.

Figure 15. Absolute speed on the stator at mid-span [m/s].

An experimental measurements campaign has been carried-out, using PIV and LDV techniques in order to obtain the instantaneous flow field in the rotor stator interaction zone, allowing a better understanding of the flow features in this zone, as well as validating the numerical computations. All the details of the experimental work are described in Ciocan (1998).

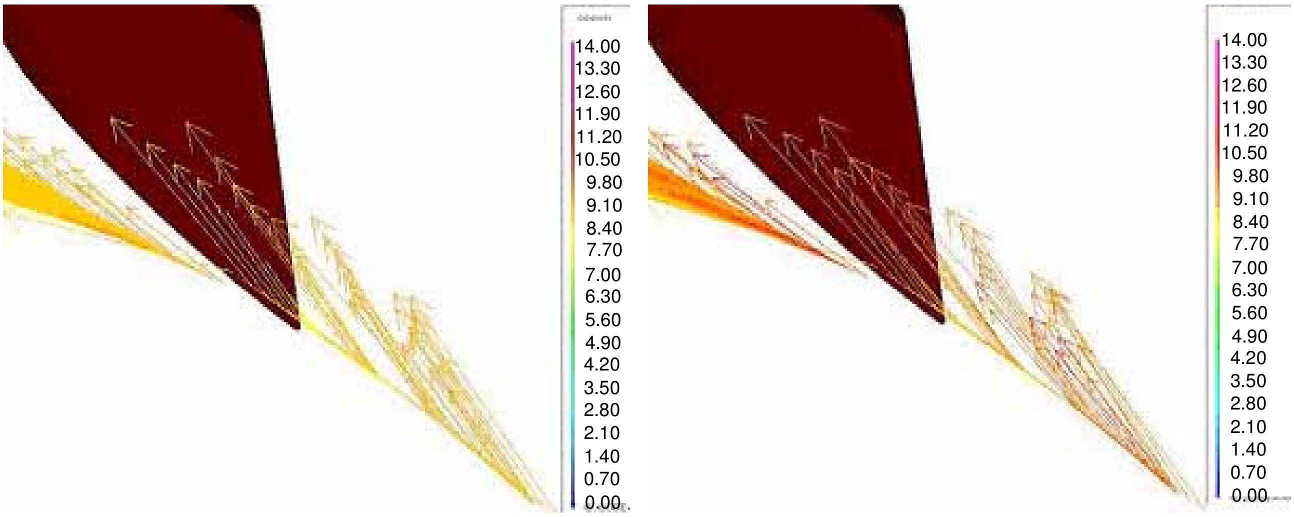
The raw results of the experimental campaign have been treated in order to represent them into the same post-processing software used to analyse the numerical results, this approach allowed a direct comparison and validation of the unsteady numerical computations.

A direct comparison of the instantaneous PIV flow field near the leading edge of the guide vanes, for two adjacent stator channels, shown in Fig. (16) A and B, to the results of the Fig. (15B) indicates that the unsteady rotor stator computation was able to accurately predict the flow behaviour at the rotor stator interaction zone, as the magnitude of the measured flow velocity and its distribution over the stator channels matches very closely the numerical predictions. A further indication of the good accuracy of the computations can be seen at Fig. (17), where the LDV measured (Fig. 17 A) and computed (Fig. 17 B) velocity vector fields near the leading edge of the guide vanes agree very closely both, in magnitude and direction.



A) Channel 1.   B) Channel 2.

Figure 16. Instantaneous PIV velocity magnitude near guide vane leading edge - 50% span; Q=Qn.

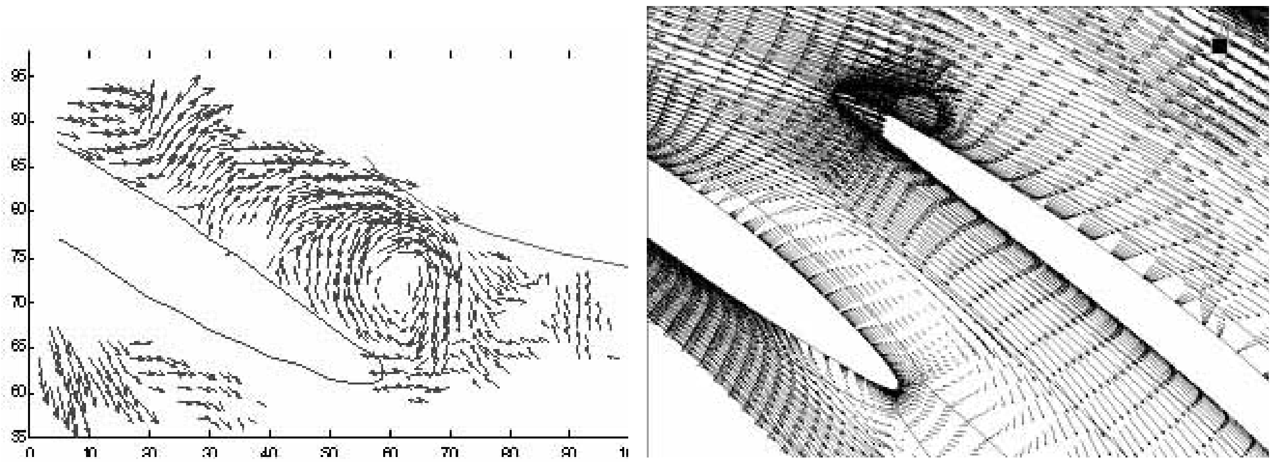


A) Measured by LDV.

B) Unsteady computation.

Figure 17. Instantaneous velocity vectors - 50% span ; Q=Qn.

The prediction of the flow features in unstable operation zone is of great importance for pump-turbine design. The experimental measurements of the flow field in the rotor-stator interaction zone shown no signs of rotating stall nor runner inlet pre-rotation for the part-load operating points considered in this study. The unstable operating zone is characterised by a strongly non uniform velocity profile, for both spanwise and tangential direction, at the outlet of the runner, as a result, for the VA Tech Pump-turbine, the hub side of the stator channel is dominated by a large flow detachment at the suction side of the guide-vanes, that occupies the entire tangential extension of the channel, originating important energy losses that could explain the drop in pump efficiency. This behaviour could be accurately predicted by the unsteady computation, as it can be seen in Fig. (18), comparing the LDV instantaneous flow field (Fig. 18A) with the corresponding numerical simulation. It should be noted that the recirculating zone predicted by numerical simulation does not occupies the entire tangential extension of the stator channel, this could be due to the larger cross sectional area of the stator channel, as it counts only 20 guide-vanes instead 22 for the original geometry. It suggests that the exact dynamic behaviour of the flow in the unstable operating zone could only be reproduced if the entire 360 deg. domain is computed.



B) Unsteady computation.

A) Measured by PIV.

Figure 18. Instantaneous velocity field near trailing edge of guide vanes at 10% span – Q=0.83Qn.

8. Conclusions

A detailed study of the rotor stator capabilities implemented into Fine Turbo and CFX-TASCflow CFD codes has been performed. According to the steady and unsteady test cases, it was shown that the unsteady rotor stator interface models implemented in both codes are capable to accurately represent the different flow features that are founded in the frame of uncompressible turbomachines. All rotor-stator interaction effects: potential, viscous or pure advective ones can be expected to be reproduced by the unsteady R-S model.

It was demonstrated that the sliding surface approach adopted by both commercial codes is very demanding in terms of mesh and time step sizes, leading to computation set-ups with several millions of nodes, even for simple geometries, in order to be able to capture small scale effects, like viscous wakes. In other hand, if the main flow phenomena are large scale ones, like the velocity field non uniformities in the runner outlet of the pump-turbine, this approach can be used for accurately simulate the flow field with reasonable meshes sizes.

The analysis of the pump-turbine behaviour showed that the flow field leaving the runner is highly non-uniform, both in hub to shroud direction and over the tangential direction at the outlet of the runner, even for operating conditions close to the nominal point. As a result, for off-design operating points, the flow entering one channel of the stator is strongly influenced by the rotor stator relative position, and even for an given channel, one can find well adapted flow over the leading edge of the guide vane near the shroud surface and bad adapted flow near the hub surface. This behaviour of the flow field in the rotor-stator zone leads to a complex flow pattern into the stator channels, which cannot be reproduced accurately by steady state rotor-stator interaction models. The unsteady computations showed good accuracy, when compared to experimental results, displaying qualitatively the same flow features found by the measurements and indicating the way the instability of machine is induced. In spite of that fact, the exact behaviour of the machine could not be predicted in the instability zone, this limitation can be linked to geometry modification applied to the stator domain (20 channels instead of 22) and the use of the periodicity boundary conditions. If one were interested on predicting the exact unstable behaviour of the machine, a full 360 deg. Computations would be necessary.

9. Acknowledgement

The authors wish to acknowledge the support by the European Community and the Swiss Federal Office for Education and Science in the frame of the ESPRIT project HPNURSA. The CFD Quality and Trust Network, QNET, also supports this initiative.

10. References

Ciocan, Gabriel Dan "Contribution à l'analyse des écoulements 3D complexes en turbomachines" Phd Thesis, INPG, Grenoble, 1998

Ciocan, Gabriel Dan, Avellan, François, Kueny, Jean-Louis, Optical Measurement Techniques for Experimental Analysis of Hydraulic Turbines Rotor-Stator Interaction, Proceedings (CDRom) of ASME Fluids Engineering Division Summer Meeting, June 11-15 2000, Boston, Massachusetts, USA

Dong R., Chu S., Katz J. "Effect of modification to tongue and impeller geometry on unsteady flow, pressure fluctuations, and noise in a centrifugal pump" Journal of Turbomachinery, vol. 119, p.506-515, 1997

Eisele K., Muggli F., Zhang Z., Casey M.V., "Ecoulement instationnaire dans un diffudeur aubé d'une pompe centrifuge", La Houille Blanche no. 3/4, p. 23-30, mai 1998

Henry Pierre Turbomachines Hydrauliques, Lausanne: Presses Polytechniques et Universitaires Romandes, 1992

Kaupert K., Holbein P., Staubli T., "A first analysis of the flow field hysteresis in a pump impeller", Journal of Fluid Engineering, vol. 118, p. 685-691, december 1996

Kaupert K., Staubli T., "Transient hysteresis in a centrifugal pump characteristic and associated unsteady flow field", ASME Fluid Engineering Division Summer Meeting, USA, 22-26 June 1997

Kuntz, M., Garcin, H., Guedes, A., Menter, F., Parkinson, E., Kueny, J.L., 2000, “Developments of CFX-TASCflow for unsteady rotor-stator simulations in the frame of the European project HPNURSA”, CFX Users Conference 2000, Munich, Germany

Toussaint M., Hureau F., Lapray J-F, "Influence des diffuseurs aubés sur le fonctionnement des pompes centrifuges ", La Houille Blanche no. 3/4, p.38-44, mai 1998

Ubaldi M., Zunino P., Barigozzi G., Cattanei A., "An experimental investigation of stator induced unsteadiness on centrifugal impeller outflow", Journal of Turbomachinery, vol. 118, p. 41-54,January 1996