

UNSTEADY VELOCITY PIV MEASUREMENTS AND 3D NUMERICAL CALCULATION COMPARISONS INSIDE THE IMPELLER OF A RADIAL PUMP MODEL

*G. Cavazzini** – *P. Dupont*** – *A. Dazin**** – *G. Pavesi** – *A.C.Bayeul**** –

*G. Bois****

* Department of Industrial Engineering, University of Padova, Italy.

**LML, UMR CNRS 8107, Ecole Centrale de Lille,

Cité Scientifique- BP48 59651 Villeneuve d'Ascq Cédex, France.

***LML, UMR CNRS 8107, Arts et Metiers PARISTECH

8, Boulevard Louis XIV 59000 Lille, France.

gerard.bois@ensam.eu

ABSTRACT

PIV measurements were performed at mid hub to shroud section inside the impeller of a vaned diffuser pump model working with air. Several previous papers have already presented part of impeller flow characteristics mainly for vaneless diffuser and near nominal mass flow rate. This paper concerns the pump configuration where the diffuser blades interacted with the impeller flow. Each PIV measuring plane was related to one particular impeller blade to blade channel and analyzed according to different relative positions of the vaned diffuser. A fully unsteady calculation of the whole pump has been performed and comparisons between numerical and experimental results are presented and discussed for four different mass flow rates. The present analysis is restricted to the outlet section of the impeller blade to blade passage for one particular impeller blade position relative to the diffuser.

NOMENCLATURE

B	impeller and diffuser width	(m)
g	acceleration	(m/s ²)
h	pump head	(m)
N	speed of rotation	(rpm)
Q	volume flow rate	(m ³ /s)
R	radius	(m)
V _r , V _u	radial and tangential absolute velocity component respectively	(m/s)
U ₂	peripheral velocity	(m/s)
Z	number of impeller blades	(no units)
β _b	blade angle	(°)
θ	azimuthal position	(°)
$psi = \frac{gh_t}{U_2^2}$	head coefficient	(no units)

Subscript

b	blade	1	impeller inlet section
i	pump inlet	2	impeller outlet section
n	impeller design operating condition	*	non dimensional quantities
o	pump outlet		
th	theoretical		

INTRODUCTION

The flow in a radial pump is highly three-dimensional, spatially non-uniform and intrinsically unsteady. Departure from axisymmetric or periodic assumptions was strongly highlighted by previous works such as the one from Sinha and Katz (2000). The diffuser vanes were found to play the major role in establishing the circumferential flow fields near the exit of the impeller as shown by several studies (Guleren and Pinarbasi, 2004; Hong and Kang, 2004; Guo and Maruta, 2005; Majidi, 2005; Rodriguez et al., 2007; Pavese et al., 2008; Feng et al., 2010; Cavazzini et al., 2011b).

This rotor-stator interaction results in both upstream and downstream flow variations in time and space, which generate noise, vibrations and unfavourable characteristics to pump performance even at design conditions. It is of primary importance to improve the knowledge of unsteady effects and rotor-stator interactions to develop design procedure able to control these undesirable aspects and to design pumps and to attain a more reliable and quiet operation.

With nowadays computer performances and dedicated codes, numerical simulations of flow inside the impeller and diffuser are proposed with reasonable precision, providing an insight look at the flow development through the pump.

These numerical results must be compared with experimental ones in order to be validated. Among the several experimental techniques that have been used, PIV measurements may provide reliable data for discussion and validation. This technique has been used in a special pump model working with air on which several papers have been already published especially by Wuibaut et al (2006), Wuibaut et al (2004) and Cavazzini et al (2009). More recently, Atif et al. (2011) and Cavazzini et al (2011a) have presented a comparison between PIV results and both frozen rotor and unsteady numerical approaches inside the outer part of the impeller near impeller design flow rate, demonstrating that frozen rotor technique is able to capture part of the real flow characteristics and can be used to get overall performances near design point only with reliable mesh configurations.

The present paper is focused on the analysis of the flow field at the mid hub to shroud section for different flow rates. PIV results inside the impeller blade to blade passages were compared with numerical results obtained with a full unsteady calculation for a particular blade position relative to the vaned diffuser.

EXPERIMENTAL SET-UP

Tests have been performed in air with the so-called SHF impeller, on a test rig specially adapted to study impeller-diffuser interactions. No volute has been placed downstream of the vaned diffuser to avoid further interactions. Outlet pressure is at atmospheric conditions; so test rig total pressure is always below atmospheric pressure. Due to this specific arrangement, the experimental set up allows flow leakage between the fixed and rotating parts of the pump model at the impeller outlet. Due to low pressure, this leakage may enter the inlet section of the vaned diffuser depending on the operating conditions.

PIV measuring planes have been obtained by combining two laser sheets in order to enlighten, at least, two consecutive impeller passages at the impeller outlet section. PIV fields have been simultaneously measured with two cameras in the impeller outlet zone and within the diffuser (see Fig. 2). The experimental procedure allowed the determination of the mean absolute velocities in two exposure frames, for up to 9 different impeller angular positions relative to diffuser vanes. Measurements have been made at mid span. More details on PIV techniques used, experimental setup and result interpretations can be found in Wuibaut et al. (2004).

The main features of the components and specifications of the considered operating condition are summarized in Table 1. As it can be noticed, to improve the pump performance at low flow rates, the impeller design flow rate ($Q/Q_n = 1$) was fixed different from the diffuser design flow rate ($Q/Q_n = 0.774$).

Fig. 1 shows the impeller theoretical head coefficient evolution versus non dimensional mass flow rate, already published by Barrand et al (1985). The curve corresponding to the measured isentropic head coefficient was obtained by torque measurements from which leakage and disk

losses have been suppressed.

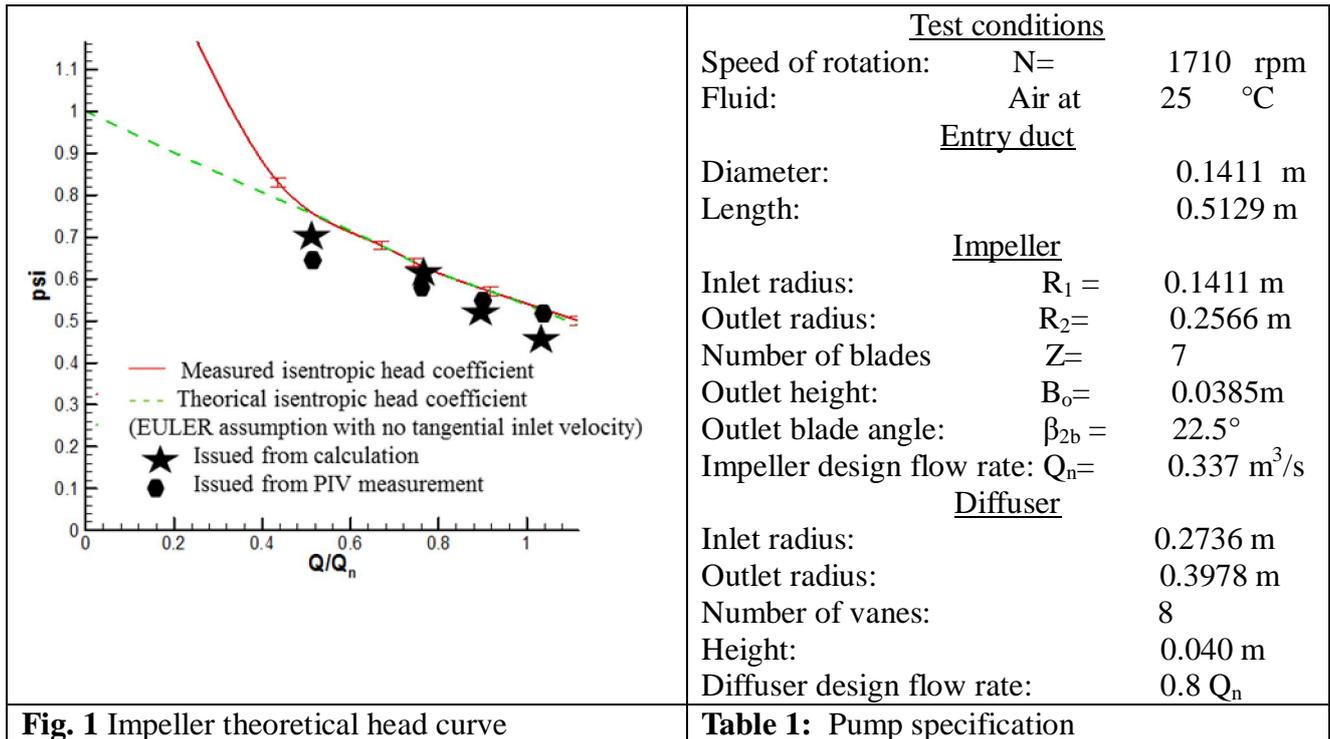


Fig. 1 Impeller theoretical head curve

Table 1: Pump specification

NUMERICAL PROCEDURE

Computations were performed on the entire pump, comprehensive of the leakage system, using the commercial software package CFX 14.0.

A fully-turbulent boundary layer was assumed on both blades and wall surfaces. The turbulence was modelled by the Detached Eddy Simulation Model (DES), which combines features of classical RANS formulations with elements of Large Eddy Simulation (LES) methods. It is based on the idea of covering the boundary layer by a Shear Stress Transport k-ε model and switching the model to the Smagorinsky-Lilly model in detached regions. The LES model was utilized in order to take advantage of its ability to provide information on turbulent flow structures and spectral distribution, which might be important to predict noise or vibrations due to stator-rotor interaction.

An unsteady model was used for all the computations. For the interface between stator/rotor blocks the standard transient sliding interface approach was chosen. For the discretization in time, a second order dual time stepping scheme was adopted. The time step for the explicit scheme Δt was chosen according to a rotation of the runner of about one degree resulting in a maximum Courant Number far from the impeller of about CFL = V*Δt/Δx = 3 and below 1 inside the impeller, where V is the velocity of the mass flow in the control volume Δx. The maximum number of iterations for each time step was set to 5, resulting in a mass residues of 10⁻⁶, momentum residues of 10⁻⁴, turbulence kinetic energy and energy dissipation residues of 10⁻⁴.

An H-type grid was used for the impeller, whereas an O-type grid was adopted for the diffuser. The leakage from the labyrinth seal was also considered and several H-blocks were built to describe the cavities. The grid was globally of 3.9•10⁶ points, with y+ values below 60 in the whole computational region and below 20 in the impeller.

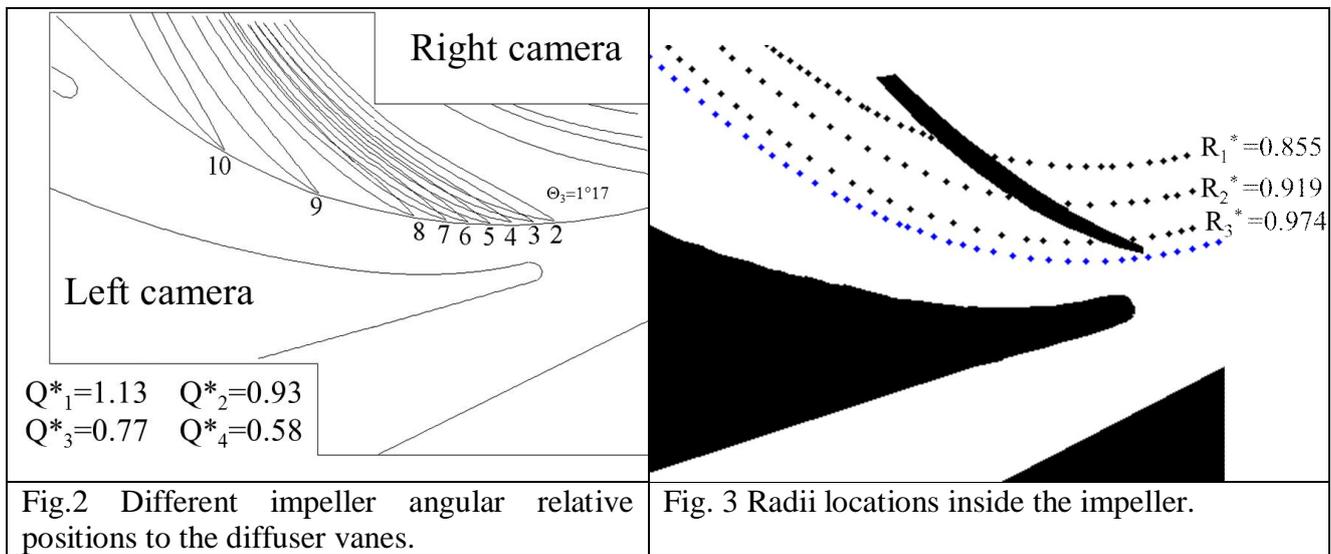
As regards the boundary conditions, mass flow rates obtained from the experimental data were prescribed at the inlet boundary and at the labyrinth close to the impeller inlet with stochastic fluctuations of the velocities with 5% free-stream turbulence intensity. At the impeller outlet the leakage mass flow rate was controlled by the known pressure in the large plenums upstream the labyrinth. The surfaces were supposed as adiabatic walls with a no-slip condition. As regards the exit boundary conditions, the experimental pressure level was prescribed as average pressure at the

diffuser outlet.

EXPERIMENTAL RESULTS

PIV images were taken for nine different impeller blade positions, whose angular reference with respect to the radial line passing at the diffuser leading edge are reported in Fig. 2. The chosen blade position 3 for which measurements and calculations are compared is shown in gray on this same figure. This corresponds to the configuration with the maximum blockage effect in the vaneless part of the diffuser. The following vaned diffuser blade leading edge is facing the outer part of the impeller blade. This relative position may lead to an interaction with impeller pressure side.

PIV results were extracted from the data base by a bi-linear interpolation method used in Wuibaut's work (Wuibaut et al (2001)) on three chosen radii located near the outlet part of the impeller at mid-span (figure 3) and are compared with the numerical results. The velocities were measured with a relative accuracy of about 2.2% for the lower flow rate and 1.2% for the larger flow rate.



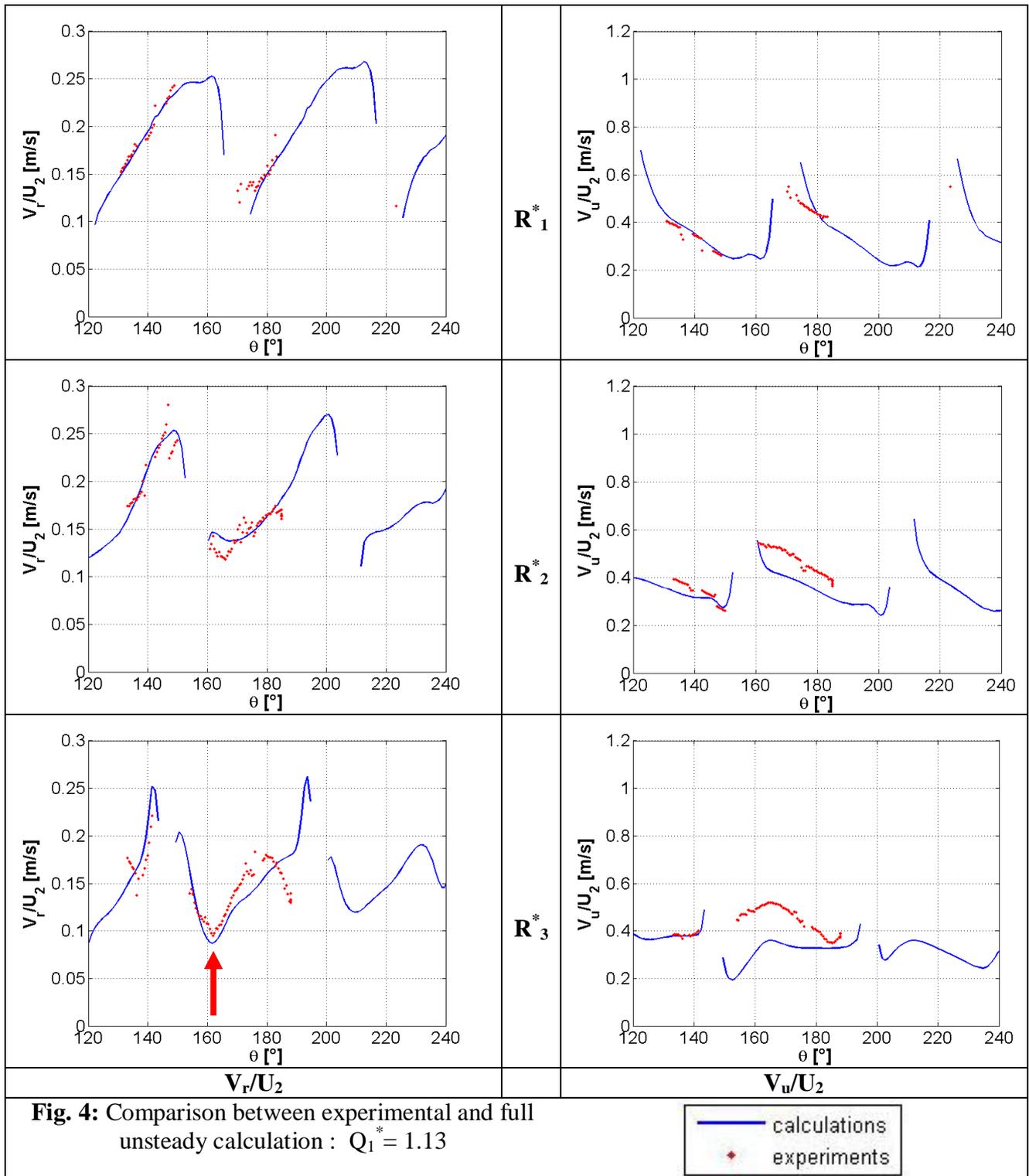
RESULTS PRESENTATION AND DISCUSSION

Four sets of figures, from figures (4) to (7), corresponding to decreasing relative mass flow rate values show PIV results and their comparisons with numerical calculations. Each set presents non dimensional radial (left column) and tangential (right column) absolute velocity components, respectively for the chosen three radial locations. Increasing values of θ corresponds to the direction of impeller rotation.

Local distributions of velocity components show that calculated blade to blade velocity evolutions are different for each blade passages. This confirms the non periodic flow behaviour inside the impeller due to the different blade numbers between impeller and diffuser and the related rotor stator interaction.

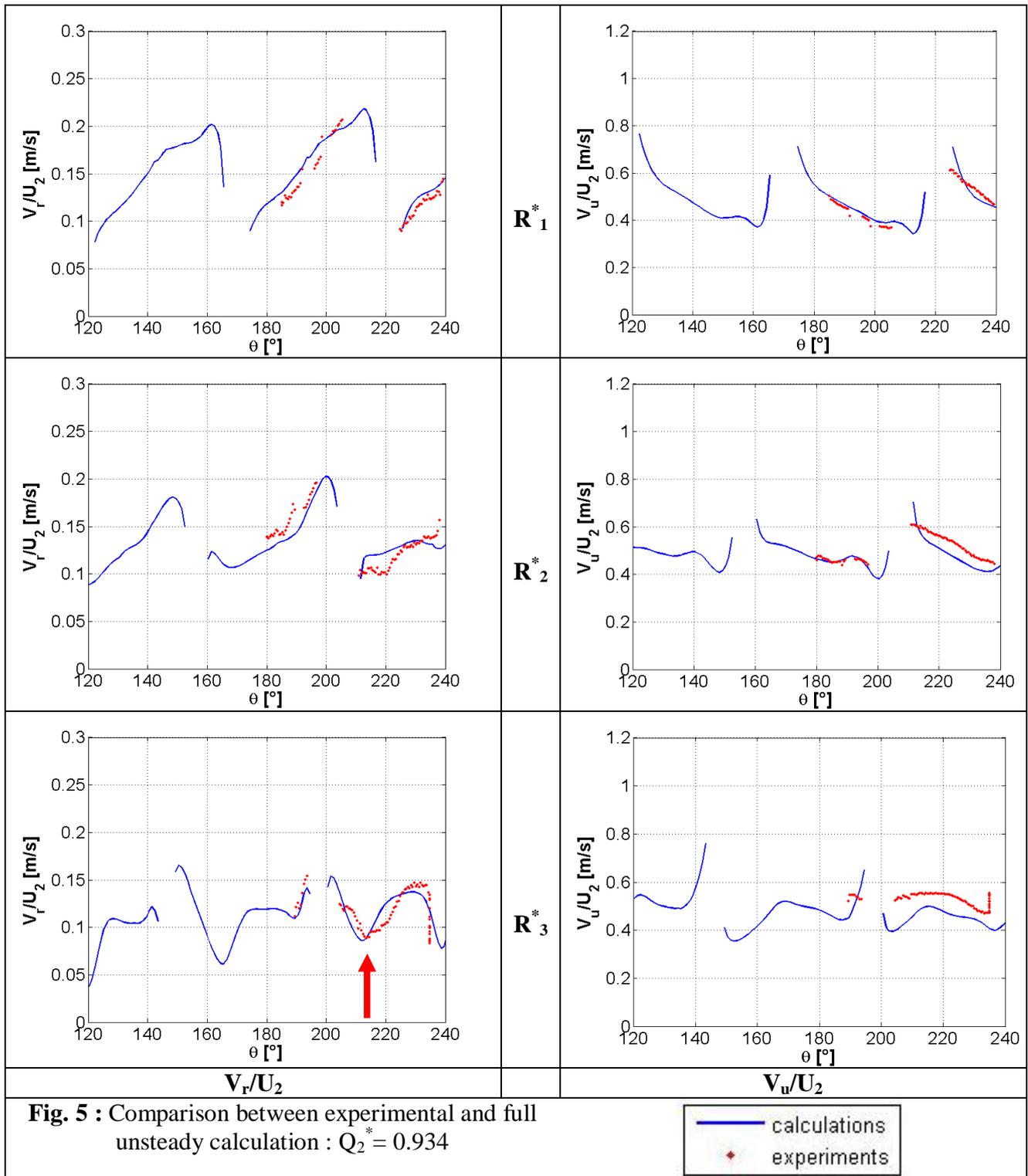
This is mainly true at off design conditions with increasing values of radius. The upstream influence of the diffuser leading edge blade is well captured by both numerical and experimental results and is much detectable looking at radial velocity components. There is a minimum radial velocity corresponding to a blockage effect, which is more pronounced for increasing values of the radius and reach the middle part of the impeller passage. This minimum velocity location, located by an arrow in each figure representing radial velocity distributions near impeller outlet, slightly move towards the impeller pressure side for decreasing mass flow rates; this is probably due to the fact that the upstream influence of the diffuser blade is transported with lower values of absolute flow angle when mass flow rate becomes lower.

Whatever the mass flow is, comparison between experiments and calculations is quite good.



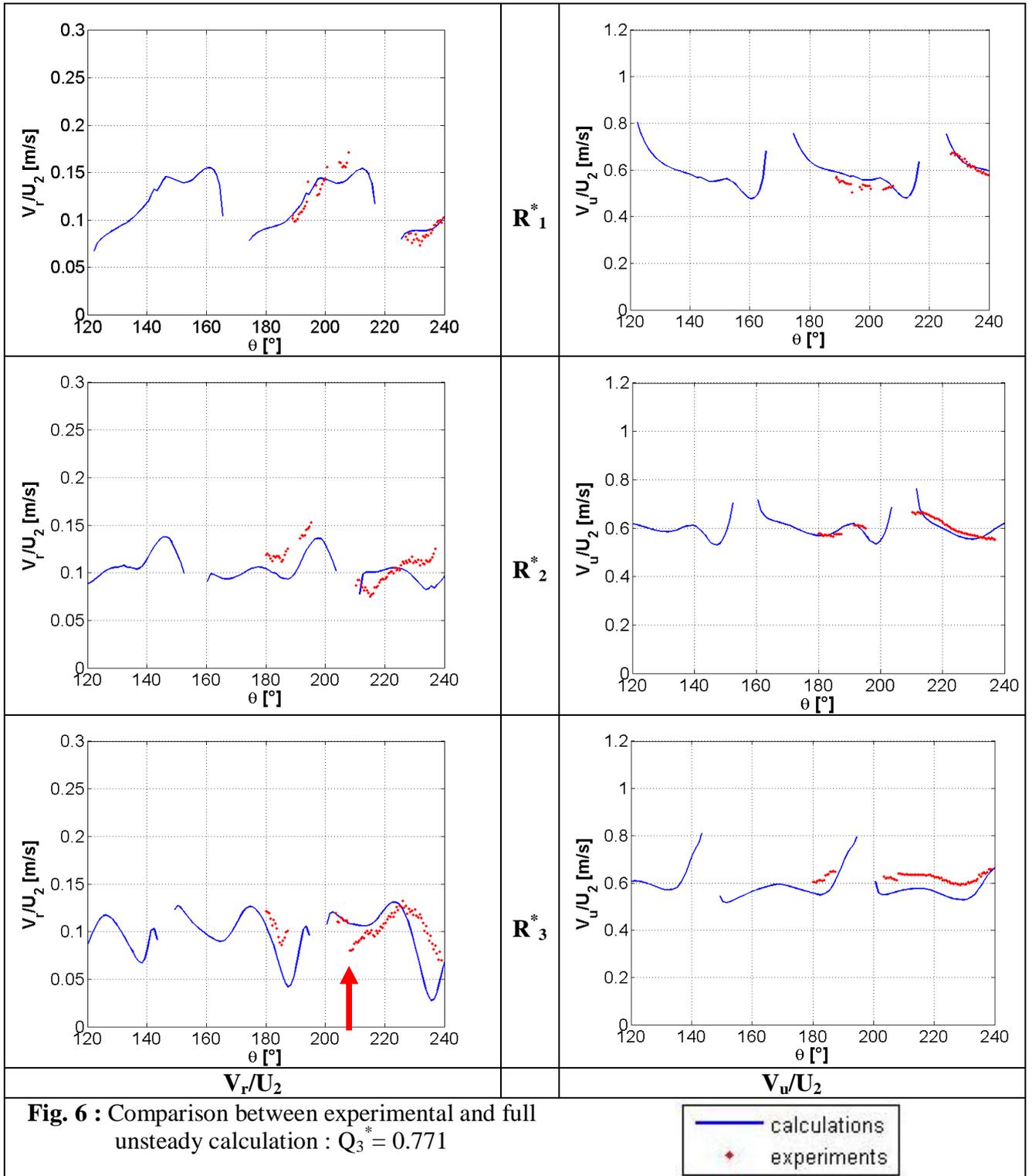
Remarkable good comparisons are present for the two first radii R_1^* and R_2^* , whereas some discrepancies can be seen, mainly on the tangential velocity component at the outer radius. For this last position, numerical results are always lower than the experimental ones, the difference becoming greater with decreasing mass flow rates.

As regards possible error sources, significant hub to shroud gradients of both the radial and tangential velocity were highlighted by the unsteady numerical analyses mainly at partial flow rates (figs. 8-9). Due to the presence of these gradients a small error in the PIV measuring plane position could partially explain the systematic differences between numerical and experimental results. Zones of significant axial velocity, due to secondary flows or to leakage flow effects, could also



affect the PIV results (fig. 10). However, it can be noticed that the moment of momentum obtained by the experimental data at impeller mid span outlet section quite well agree with the impeller theoretical head, given in figure (1).

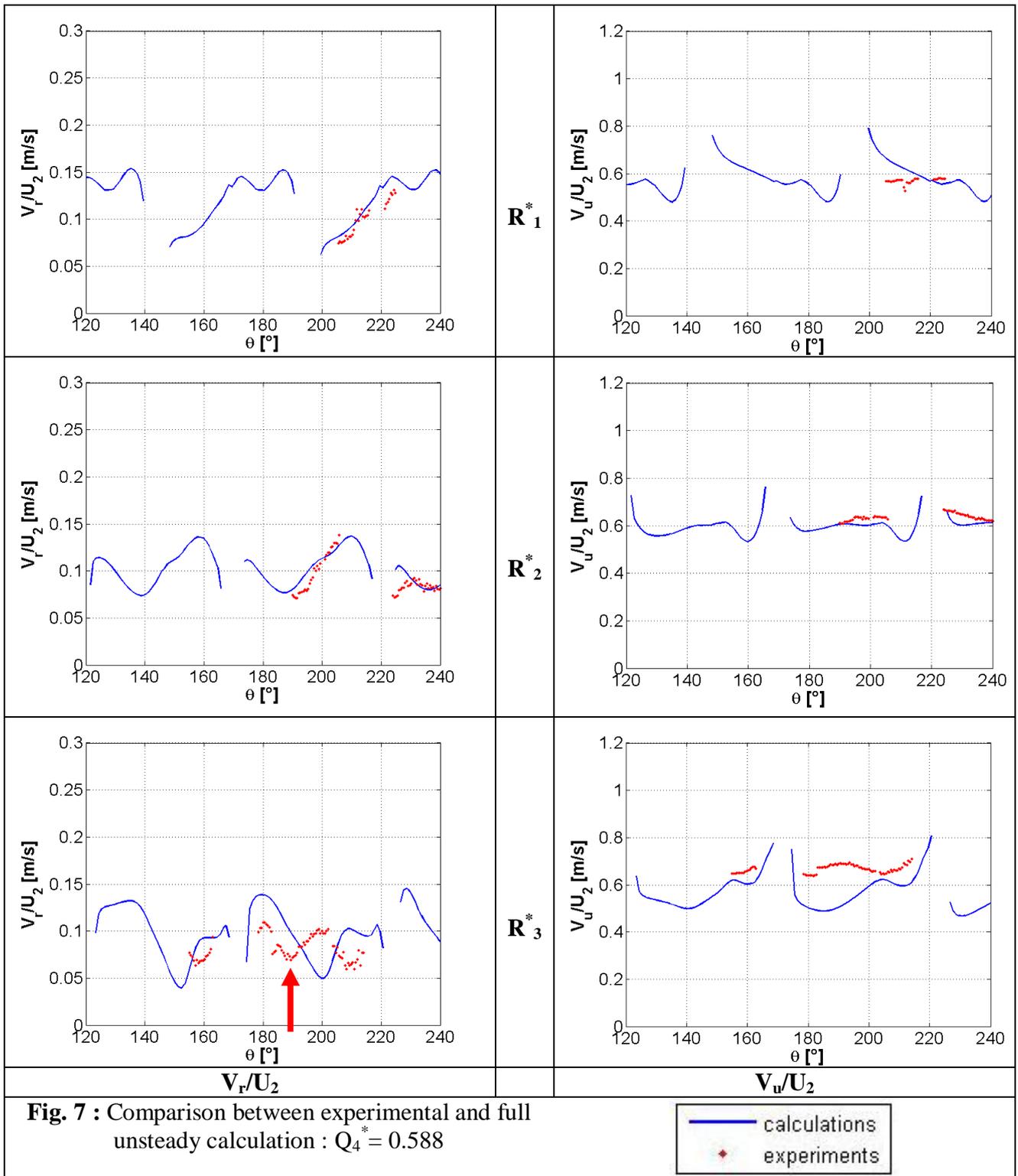
Of course, this argument is only valid under the assumption that mid span results correctly represent overall theoretical performances over the entire hub to shroud section for all impeller passages. In this respect, the differences between numerical and experimental tangential velocity levels may be due to the numerical approach and closure turbulence models. The fact that the radial component is well predicted, both in levels and gradients is consistent with mass flow conservation. Calculated boundary layers and jet and wake structure developments including rotor stator



interactions may be different than the real experimental ones. This has to be checked more deeply both with initial inlet flow conditions in front of the impeller.

Another reason, which is difficult to evaluate, concerns the leakage flow effects, already pointed out, occurring between impeller outlet and vaned diffuser section. If this effect is considered to be an important one, it will lead to lower the tangential velocity component at impeller outlet section. This assumption may be valid, because results plotted on figure 1 show that the difference between experiments and calculations is increasing for decreasing mass flow rates. This is coherent with increasing leakage amount when mass flow decreases.

More detailed experimental investigations of this particular point are in progress. On the other



hand, numerical calculations are going to be performed without leakage in order to get an evaluation of its effects.

CONCLUSIONS

The flow variations in time and space due to the rotor-stator interaction affects the pump performance even at design conditions, generating noise, vibrations and unfavourable characteristics. A more in depth knowledge of these unsteady effects is hence of primary importance since it will allow to develop optimized design criteria able to control these undesirable aspects so as to enlarge the pump working range with more stable and quite operating conditions.

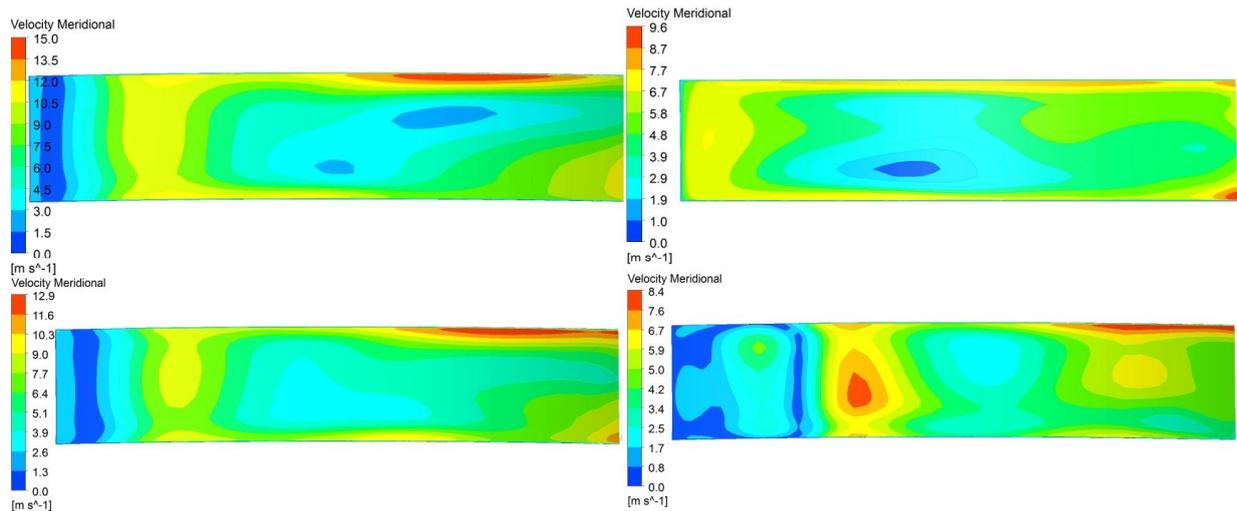


Fig. 8 Radial velocity component at radius R_3^* for the four analysed operating conditions

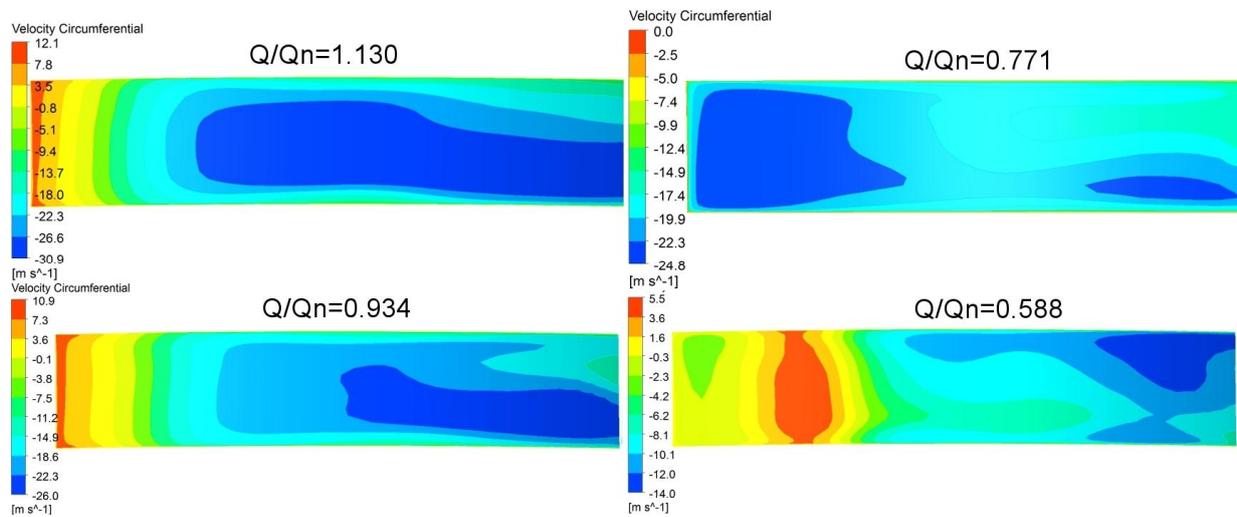


Fig. 9 Tangential velocity component at radius R_3^* for the four analysed operating conditions

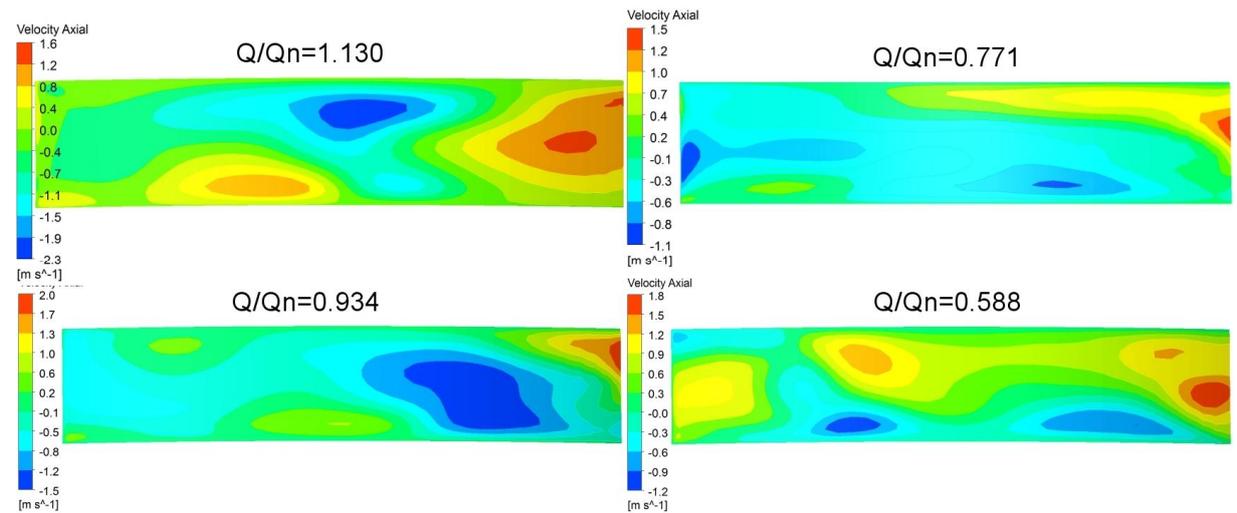


Fig. 10 Axial velocity component at radius R_3^* for the four analysed operating conditions

The paper provided an in-depth analysis of the rotor-stator interaction in the pump by means of an optical PIV experimental analysis combined with an unsteady numerical approach. Unsteady numerical profiles and PIV experimental blade to blade distributions inside a centrifugal impeller were compared at one particular impeller blade position relative to the vaned diffuser in order to detect and evaluate rotor-stator interactions for a wide range of mass flow rates.

Results are presented at the mid section between hub and shroud in the outer part of the impeller blade to blade passage. These interactions are well captured and predicted for all mass flow rates by the DES turbulence model, especially for the radial velocity component inside the impeller. Tangential velocities are also well predicted with the exception of the last radial position, close to impeller outlet section, where numerical results are always below experimental ones. These differences appear to be more pronounced for decreasing mass flow rates. Further experimental and numerical investigations are going on in order to explain the reasons of these differences in order to evaluate leakage flow possible effects on one hand and numerical ones on the other hand.

REFERENCES

- Atif A., Benmandsour S., Bois G., Dupont P. (2011), "Numerical and experimental comparison of the vaned diffuser interaction inside the impeller velocity field of a centrifugal impeller". *Science China*. N°1. Vol. 54. January. pp 1-9.
- Barrand J.P., Caignaert G., Graesser J.E, Rieutord.E., (1985), "Synthesis of the results of tests air and water aimed at critical re-circulating flow rates detections on the inlet and outlet of a centrifugal impeller", *La Houille Blanche*, N°5.
- Cavazzini G., Dupont P., Pavesi G., Dazin A., Bois G., Atif A., Cherdieu P. (2011a): "Analysis of unsteady Flow velocity fields inside the impeller of a radial flow pump : PIV measurements and numerical calculation comparisons", *Proceedings of ASME-JSME-KSME Joint Fluids Engineering Conference*, July 24-29, 2011, Hamamatsu, Shizuoka, Japan.
- Cavazzini G., Pavesi G., Ardizzon G. (2011b): "Pressure Instabilities in a Vaned Centrifugal Pump", *Proceedings of the Institution of Mechanical Engineers, Part A, Journal of Power and Energy*, vol. 225(7), pp. 930-939
- Cavazzini G., Pavesi G., Ardizzon G., Dupont P., Coudert S., Caignaert G., Bois G. (2009): "Analysis of the Rotor-Stator Interaction in a Radial Flow Pump", *La Houille Blanche, Revue Internationale de l'eau*, n. 5/2009, DOI 10.1051/lhb/2009067, ISSN 0018-6368 (Print) 1958-5551 (Online), pp. 141-151.
- Feng J., Benra F K. Dohmen H J. (2010) "Unsteady flow investigation in rotor-stator interface of a radial diffuser pump" *Forschung in Ingenieurwesen*, vol. 74(4) pp. 233-242
- Guleren, K. M., Pinarbasi, A. (2004) "Numerical simulation of the stalled flow within a vaned centrifugal pump" *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, vol. 218, pp. 425–435.
- Guo, S., Maruta, Y. (2005) "Experimental investigations on pressure fluctuations and vibration of the impeller in a centrifugal pump with vaned diffusers." *JSME International Journal, Series B*, vol. 48(1), pp. 136–143.
- Hong, S.-S., Kang, S.-H. (2004) "Flow at the centrifugal pump impeller exit with the circumferential distortion of the outlet static pressure", *Journal of Fluids Engineering*, vol. 126(1), pp. 81–86.
- Majidi, K. (2005) "Numerical study of unsteady flow in a centrifugal pump", *Journal of Turbomachinery*, vol. 127, pp. 363–371.
- Pavesi G., Cavazzini G., Ardizzon G. (2008) "Time-frequency Characterization of the Unsteady Phenomena in a Centrifugal Pump", *International Journal of Heat and Fluid Flow*, vol. 29 (2008), pp. 1527-1540, ISSN: 0142-727X, DOI: 10.1016/j.ijheatfluidflow.2008.06.008
- Rodriguez, C. G., Egusquiza, E., Santos, I. F. (2007) "Frequencies in the vibration induced by the rotor stator interaction in a centrifugal pump turbine", *Journal of Fluids Engineering*, vol. 129(11), pp. 1428–1435.
- Sinha M., Katz J. (2000) "Quantitative Visualization of the Flow in a Centrifugal Pump with Diffuser Vanes. Part I: On Flow Structures and Turbulence." *J. Fluids Engineering*. 122: 97–107.
- Wuibaut G., Bois G., El Hajem M., Akhras A., Champagne J Y., (2006), "Optical PIV and LDV comparisons of Internal Flow Investigations in SHF Impeller." *Int. J. of Rotating Machinery*. 1-9.
- Wuibaut G., Dupont P., Caignaert G., Bois G., (2004), "Rotor Stator Interactions in a Vaned Diffuser Radial Flow Pump", *Proceedings of the 22nd IAHR Symposium on Hydraulic Machinery and Systems*. Stockholm. Sweden.
- Wuibaut G., Dupont P., Caignaert G., Stanislas M. (2001) "PIV measurements at the outlet part of a radial flow pump impeller and the vaneless diffuser in design and off design conditions", *4th International Symposium on Particle Image Velocimetry*, September 17-19, Göttingen Germany.