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

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

*G. Bois****

* Departement of Mechanical 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 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)
Q_n	impeller design volume flow rate	(m^3/s)
R	radius	(m)
Vr, Vu	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	head coefficient psi= gh_{th} / (U_2 ** 2)	(no units)

Subscript

i	inlet, impeller
o	outlet
th	theoretical
2	outlet section of the impeller
*	non dimensional quantities

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 et al (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 Feng et al (2007). 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 (2004a) et Cavazzini et al (2009). More recently, Atif et al (2011), Cavazzini et al (2011), 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.

The present paper is focused on the analysis of the flow field at the mid hub to shroud section for different flow rate. 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 absolutes 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 paper Wuibaut et al (2004b).

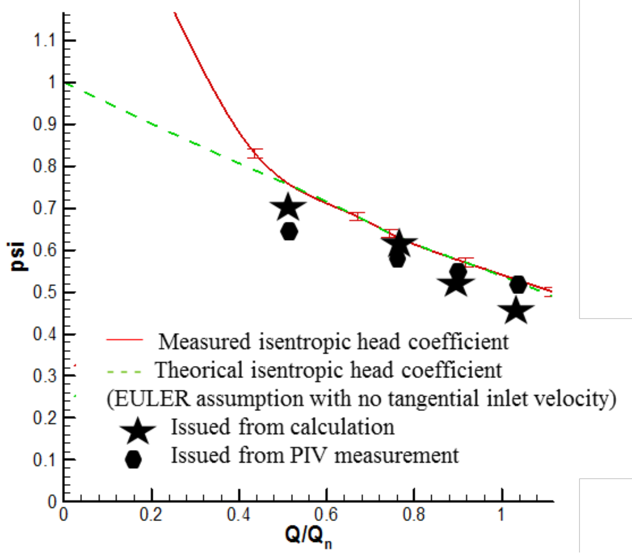


Fig. 1 Impeller theoretical head curve

Test conditions		
Speed of rotation:	N=	1710 rpm
Fluid:	Air at	25°C
Impeller		
Inlet radius:	Ri =	0.1411 m
Outlet radius:	R_o=	0.2566 m
Number of blades	Z=	7
Outlet height:	B_o=	0.0385m
Outlet blade angle:	β_{2b} =	22.5°
Impeller design flow rate:	Qn=	0.337 m^3/s
Entry duct		
Diameter:		0.1411 m
Length:		0.5129 m
Diffuser		
Inlet radius:		0.2736 m
Outlet radius:		0.3978 m
Number of vanes:		8
Height:		0.040 m
Diffuser design flow rate:		0.8 Qn

Table 1: Pump specification

The main features of the components and specifications of the considered operating condition are summarized in Table 1.The particular mass flow rate of Q/Qn = 0.774 corresponds to an off design condition for the impeller and a near design condition for the vaned diffuser. The choice of a different design condition for impeller (Q/Qn close to 1.) and diffuser was intended to improve performance at low flow rates for the whole pump.

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.

NUMERICAL PROCEDURE

Computations were performed using the commercial software package CFX 10.0.

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 Edge 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 importance 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 was chosen according to a rotation of the runner, of about one degree resulting a Courant Number of about CFL=3. 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 $3.9 \bullet 10^6$ points, with y+ values below 60 in the whole computational region.

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

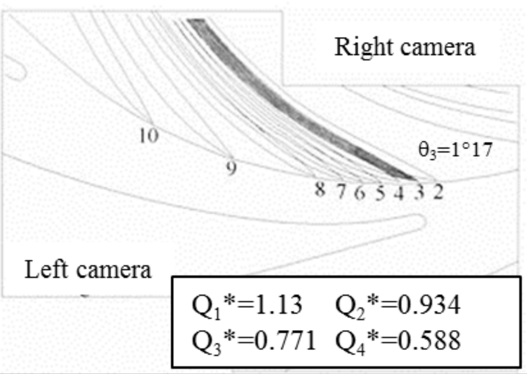


Fig.2 Different impeller angular relative position to the diffuser vanes.

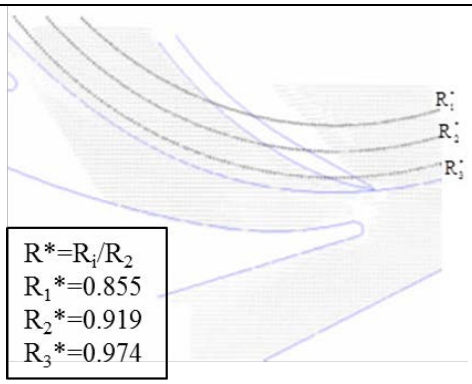


Fig. 3 Radii locations inside the impeller.

RESULTS PRESENTATION AND DISCUSSION.

Four sets of figures, from figures (4) to (7), corresponding to decreasing relative mass flow rates 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 figures 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 deceasing mass flow rates. It can be noticed that experimental tangential mean values at impeller outlet section well agree with overall experimental impeller theoretical performances, based on Euler approach, given in figure (1). Of course, this argument is only valid under the assumption that mid span results correctly represents 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 predict, 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 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 plot 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

Comparison between unsteady numerical approach and optical PIV experimental blade to blade distribution inside a centrifugal impeller has been presented for 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 captures and predicted especially for the radial velocity component for all mass flow rates. Tangential velocities are also well predicted instead 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.

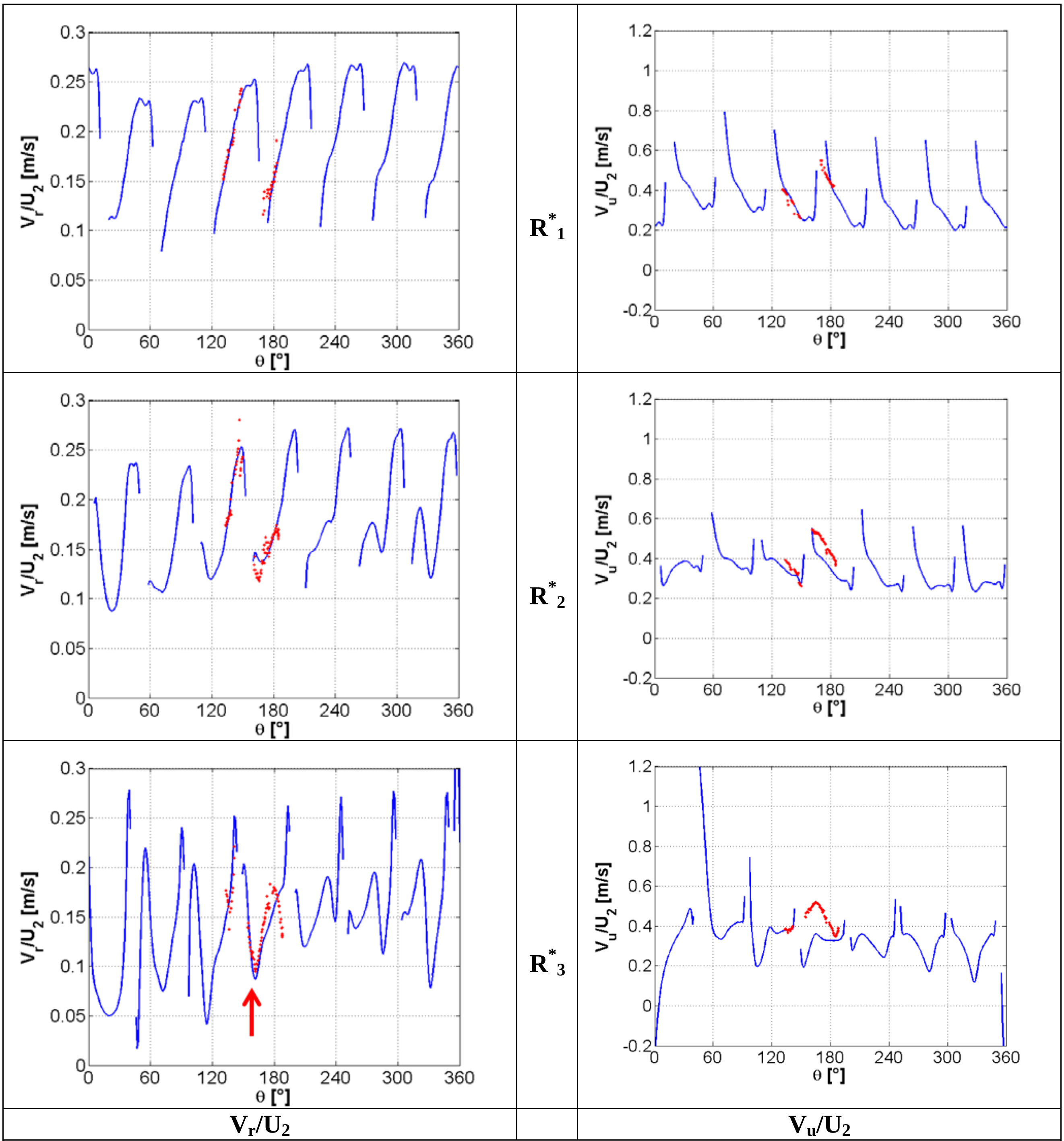
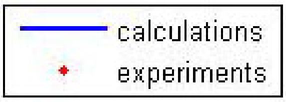


Fig. 4: Compaison between experimental and full unsteady calculation : $Q_1^* = 1.13$



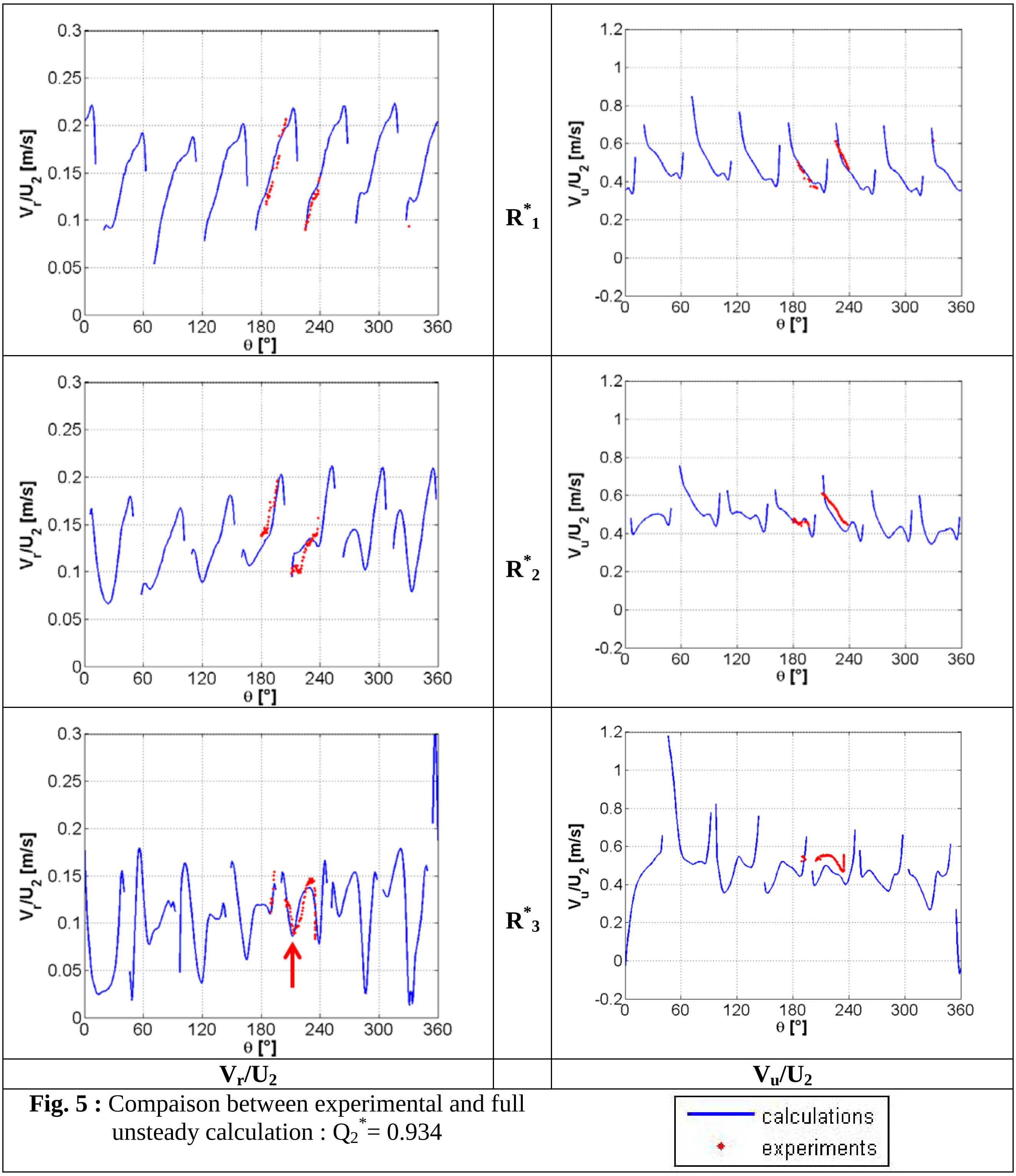


Fig. 5 : Compaison between experimental and full unsteady calculation : $Q_2^* = 0.934$

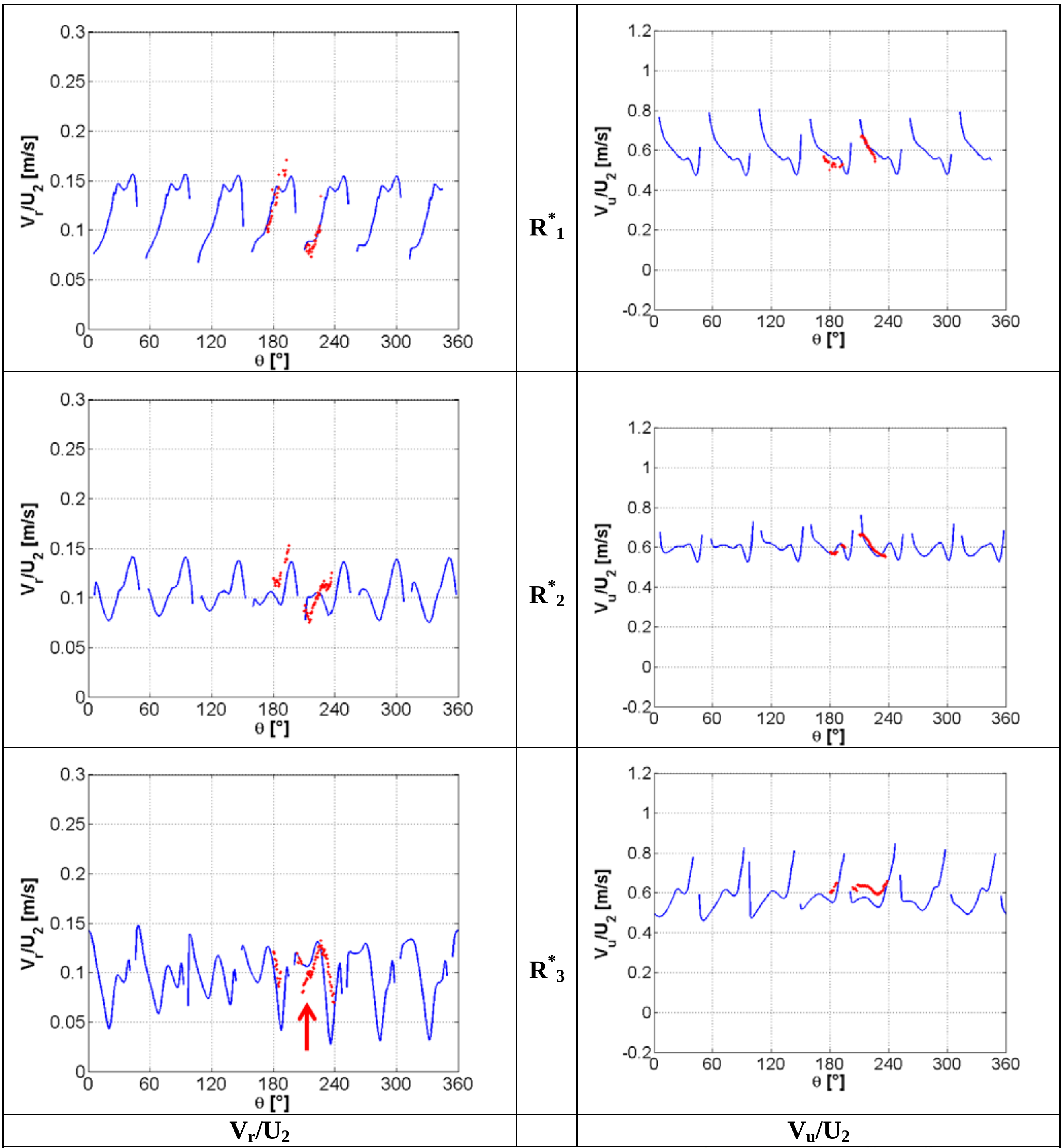
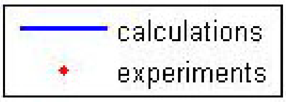


Fig. 6 : Compaison between experimental and full unsteady calculation : $Q_3^* = 0.771$



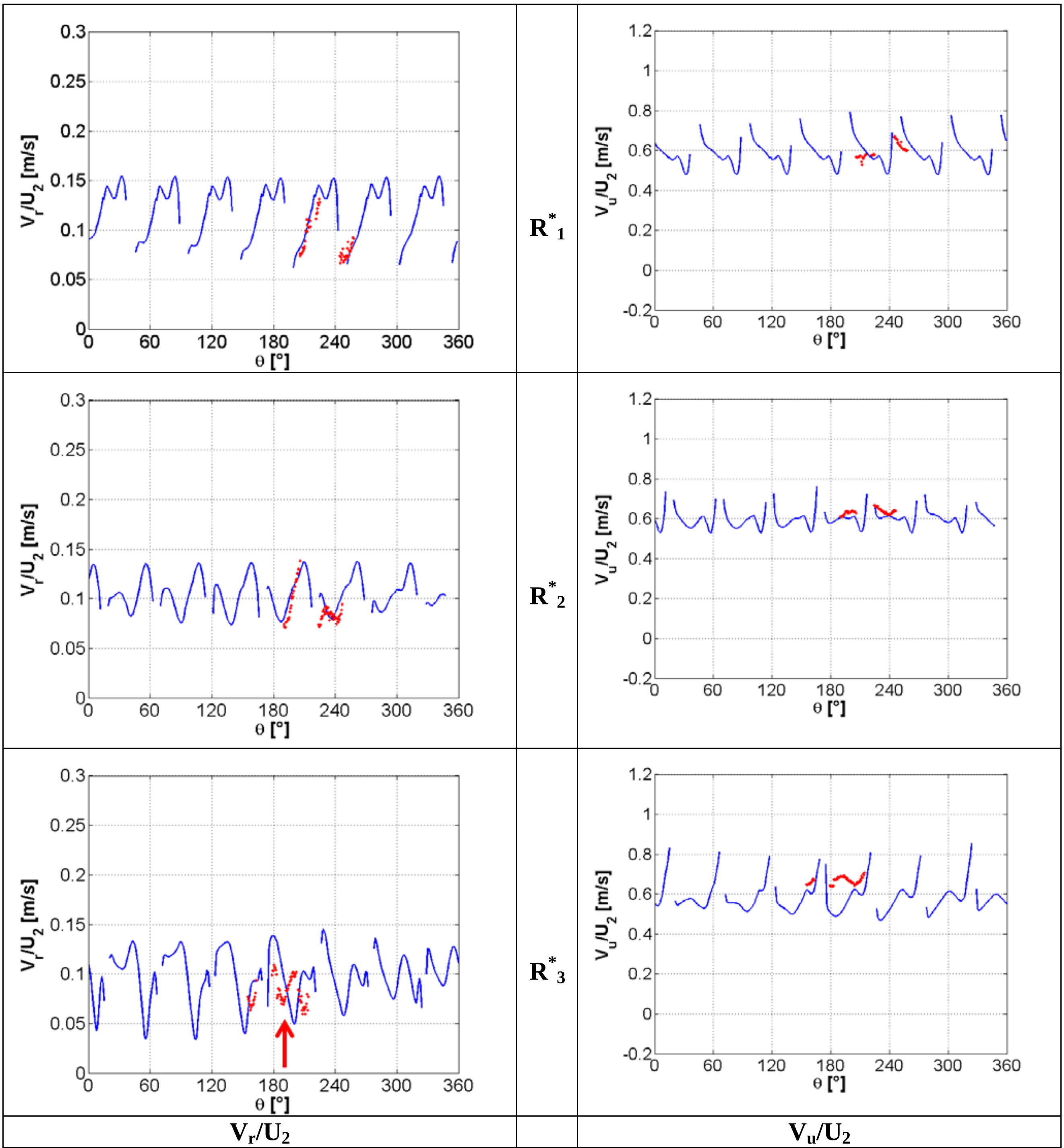
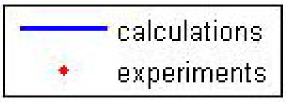


Fig. 7 : Compaison between experimental and full unsteady calculation : $Q_4^*= 0.588$



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. (2011): "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 Engeering Conference, July 24-29, 2011, Hamamatsu, Shizuoka, Japan.

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. (2007), "A Study on Impeller-Diffuser Interactions in a Radial Pump." Proceedings of the 7th European Conference on Turbo machinery. Fluid Dynamics and Thermodynamics, Athens, Greece, 773-782

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.

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.