NUMERICAL INVESTIGATIONS OF VELOCITY FLOW FIELD INSIDE A CENTRIFUGAL PUMP IMPELLER IN INTERACTIONS WITH TWO VANED DIFFUSER CONFIGURATIONS AND COMPARISON WITH PIV MEASUREMENTS

A. ATIF[a] A. – ALIOUANE[b]- A. DAZIN[c] - P. DUPONT[d] - G. BOIS[e]

[a] Laboratory of Mechanical Energetics and Conversion Systems, FGMGP/USTHB, BP32 El Alia, Alger, Algeria. *amatif22@gmail.com*
[b] Laboratory of Mechanical Energetics and Conversion Systems, FGMGP/USTHB, BP32 El Alia, Alger, Algeria. *ai.aliouane@gmail.com*
[c] Laboratory of Mechanics of Lille, Arts et Metiers PARISTECH, 8 Bd Louis XIV, Lille, France. *antoine.dazin@ensam.eu*
[d] Laboratory of Mechanics of Lille, Ecole Centrale de Lille, Cité Scientifique, Villeneuve d'Ascq, France. *patrick.dupont@centralelille.fr*
[e] Laboratory of Mechanics of Lille, Arts et Metiers PARISTECH, 8 Bd Louis XIV, Lille, France. *gerard.bois@ensam.eu*

Abstract :
The paper refers to the analysis of interactions between the impeller and the vaned diffuser of a radial flow pump tested in air. The vaned diffuser comes with two configurations: channel type and cascade type. The study deals with unsteady numerical simulations of the flow for complete 3D geometries of impeller and vaned diffusers in order to capture the full interaction effects. The task is conducted close to design operating conditions. The results focus on the flow pattern at the outlet part inside the impeller and influence from the diffuser vanes. Results from both diffuser configurations are compared and discussed. Results are also compared with the available PIV measurements.

Keywords : Centrifugal Pump – Vaned Diffuser – Impeller Diffuser interaction – Unsteady flow simulation

1 Introduction
Flow behaviour in radial flow pumps is quite complex and is strongly depending on rotor-stator interactions. Knowledge improvement of unsteady effects of the interactions has already been a favorite study theme for a large number of researchers (Feng et al. [4], Sinha and Katz [6], Cavazzini et al. [3]). A great number of rotor-stator interaction analyses based on experimental and modeling activities have been realized, but still further investigations are needed so that pump designs could benefit from controlling the undesirable aspects in order to construct them in a more compact, efficient, and quiet operation.

Impeller-vaned diffuser interactions in radial flow pumps have two aspects: fluctuating flow structure leaving the impeller impose non uniform inlet conditions to downstream flow in diffuser [4], and the proximity of diffuser vanes to impeller outlet are found to play a major role in establishing the upstream flow in the impeller [2]. While most researchers are mainly focusing on the first aspect of interaction in concern with enhancing pressure recovery in diffusers [5], less is done on the second aspect which is a source of vibrations, noise, and unfavorable characteristics to pump performance even at design conditions [2].
Two types of vaned diffusers can be identified: the first consists of thin curved blades called cascade type similar to those used in axial turbomachines, and the second is constructed of divergent channels through the use of wedged shaped or island vanes.
With the evolution of LDV and PIV non-intrusive techniques, it has been possible to measure velocity fields within impeller and diffuser of radial flow pumps and to provide reliable data for discussion and validation. The tests highlighted the strong unsteady character of this rotor-stator interactions (Wuibaut et al. [8], [9], [10], Pavesi et al. [5]).
Numerical simulation has also been carried out to analyze internal flows in pumps. To be able to capture the interaction effects by numerical simulation, it should be undertaken a full 360° study for both impeller and diffuser. Beside the geometrical and physical modeling aspects and concerns about the use of simplifications, such complex flows should be treated by fully unsteady mode to be closer to the real physics. (Atif et al. [1], Cavazzini et al. [3])
The aim of the present study is to point out the influence of both diffuser configurations on the flow structure near the outlet inside the impeller. To do so, unsteady numerical calculations are achieved and compared with available PIV measurements.

2 Experimental procedure

Experimental data used for comparison in this study are taken from the extensive PIV work made by Wuibaut [7]. The tests have been carried out in air with the so-called SHF impeller, on a test rig specially adapted to study impeller-diffuser interactions (Figure 1). The impeller is fitted during each test with one of two configurations of vaned diffuser: channel type and cascade type (Figure 2). The main geometrical characteristics of the pump, including diffusers, and operating conditions are given in Table 1. It is worth to point out that some operating conditions are not exactly the same when the impeller is fitted with channel type diffuser or cascade type; this is due to the fact that the use of the same orifice plate for setting the flow rate does not yield to exactly the same value for each configuration. Also blade position is slightly different due to PIV synchronism.

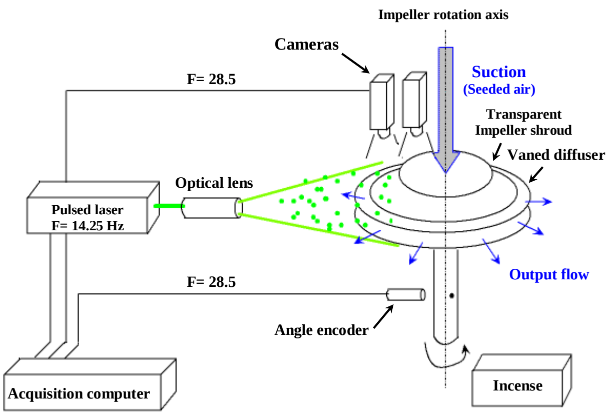
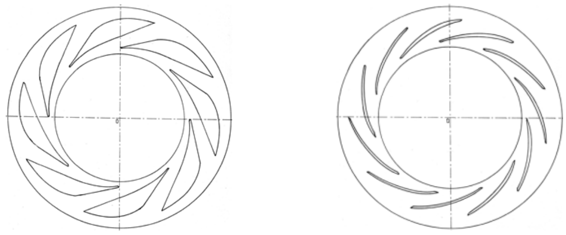


Figure 1 : Test setup



Channel type diffuser **Cascade type diffuser**

Figure 2: Vaned Diffuser Configurations

PIV fields have been measured simultaneously in two exposure frames in the impeller outlet part, only at mid span plane between hub and shroud (Figure 3). The experimental procedure allows for determination of mean absolute velocities at a prescribed impeller blade angular position relative to diffuser vanes. Further details on experimental procedure and results interpretation are described by Wuibaut [7].

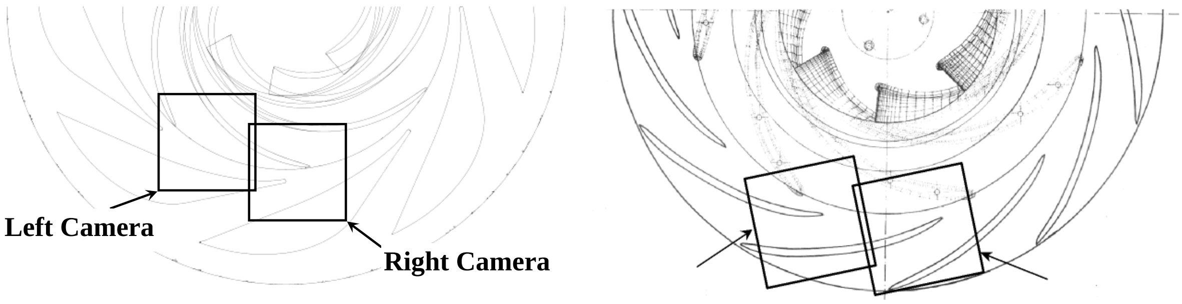


Figure 3: Location of PIV cameras

Table 1: Pump Specifications

Test conditions	
Speed of rotation	1710 rpm
Fluid	Air at 25 °C
Design flow rate	Q_n= 0.337 m^3/s
Impeller	
Inlet radius	0.1411 m
Outlet radius	R_o= 0.2566 m
Number of blades	7
Outlet height	B_o= 0.0385 m
Outlet blade angle	22.5°
Peripheral velocity	U_o= 45.95 m/s
Channel type Diffuser	
Normalized flow rate	$Q^*= Q/Q_n$= 0.934
Blade position	1.14 °

Inlet radius	0.2736 m
Outlet radius	0.3978 m
Number of vanes	8
Height	0.040 m
Cascade type Diffuser	
Normalized flow rate	$Q^* = Q/Q_n = 0.961$
Blade position	1.17 °
Inlet radius	0.2736 m
Outlet radius	0.3978 m
Number of vanes	11
Height	0.040 m

3 NUMERICAL PROCEDURE

The pump is modeled as a set of complete geometries of the impeller and the vaned diffuser (Figure 4). An entry duct long enough is added to account for inlet flow conditions. Flow leakage occurring in the gaps between rotating impeller and stationary diffusers is accounted for in the numerical study. The pump is tested at the same conditions as in the experimental procedure, at flow rates close to impeller design.

The pump is completely meshed with unstructured tetras. Tetras are more flexible elements to model complex geometries but the outcome is a larger problem dimension. Mesh refinements are placed at particular locations like blade edges, gaps, walls and interfaces. The total mesh is about 2.5 million nodes.

Computations are performed using licensed ANSYS CFX code. Three-dimensional, incompressible Reynolds averaged Navier-Stokes equations are solved. Boundary conditions are mass flow rate inlet and static pressure outlet, with automatic near wall treatment. The turbulence is simulated with k-ω based SST model. The general convergence criteria is set to momentum residues of 10^{-5}. For unsteady mode, standard transient sliding interface approach is used. A time step equivalent to a 0.5 degree rotation of the impeller is used. Final calculations are achieved after 3 turns. The results are extracted at blade position 1° which is the closest to the prescribed experimental settings.

For a more precise analysis of the influence of the diffuser vanes on the flow within the impeller, a frame of reference is set up as in Figure 5. $\boldsymbol{\theta}$, the circumferential location in degrees, is equal to 0 at the line joining the center of rotation to the diffuser main vane leading edge. Increasing $\boldsymbol{\theta}$ indicates the direction of rotation (clockwise). It sets up the pressure side and the suction side of the blades. $\boldsymbol{\varphi}$ is the angular position of the impeller main blade trailing edge with respect to diffuser vanes. It is used as prescribed positions for comparing simulations and PIV measurements. Only one angular position of the impeller blade is considered in the present study as given in Table 1.

Simulation data are extracted on three blade to blade constant radius lines as shown in Figure 6, and specifications are given in Table 2. This allows illustrating velocity profiles at the impeller outlet and comparing with experimental measurements.

All velocities are normalized with respect to impeller outlet peripheral velocity $\boldsymbol{U_0}$.

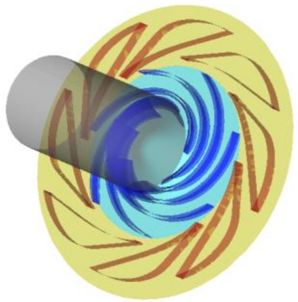


Figure 5: Pump Modeling with channel-type diffuser

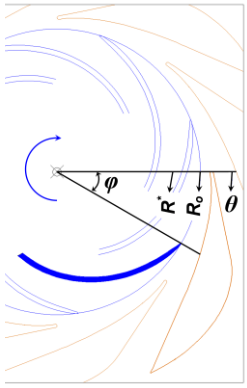


Figure 6: Frame of reference

Table 2: Normalized Radii Definition

$R^* =$	R_1^*	R_2^*	R_3^*
R/R_o	0.855	0.919	0.974

4 Results and discussion

4.1 Radial and tangential velocity contour plots at impeller outlet

Figure 7 shows blade to blade contour plots of respectively normalized radial and tangential velocity at impeller outlet for both diffuser configurations, only at mid span from hub to shroud.

As can be seen from the figure, blade to blade distributions of velocities are not similar from one impeller passage to another, predicted by unsteady calculations for both diffuser configurations. This non periodicity of the flow field confirms the strong interaction with the diffuser vanes and their effects even far upstream inside the impeller.

Differences do exist between channel type and cascade type predictions, that is :

For C_r^*, high blockage zones in front of diffuser vane leading edges and strong gradient of velocity from pressure side to suction side of the impeller blades are more pronounced for cascade type diffuser than for channel type. The number of spots is equal to the number of diffuser vanes.

For C_u^*, it is observed an increasing tangential velocity towards the impeller outlet, uniformly from pressure side to suction side of the impeller blades, with smoother variation in the case of channel type diffuser, but rather more distorted flow at exit in case of cascade type diffuser.

These observations gives in rather a more distorted flow field in the outlet part of the impeller in case of cascade type diffuser than for channel type. This suggests that channel type diffuser impose a smoother flow structure on the flow inside the impeller and the number of diffuser vanes has more impact than does their shape.

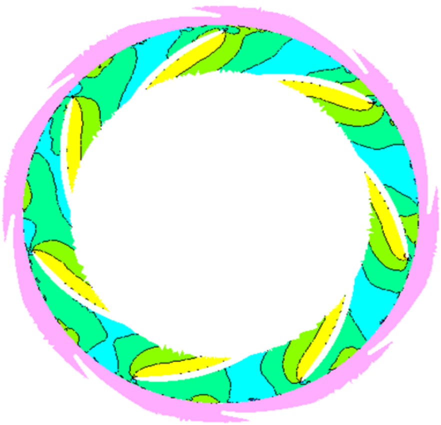
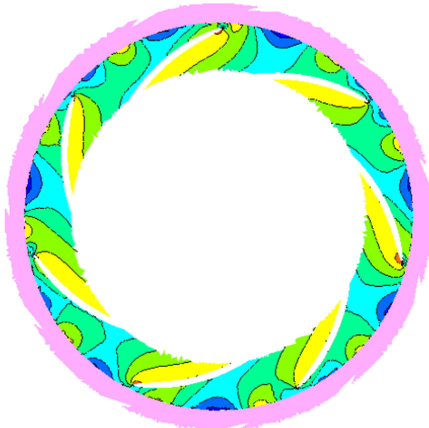
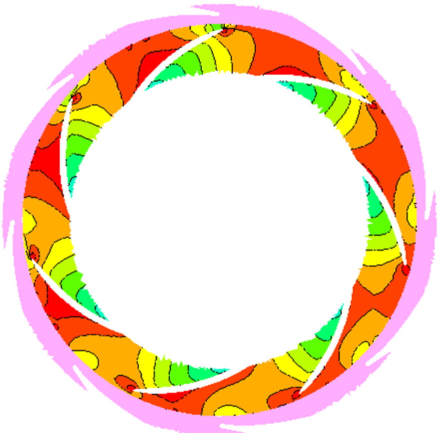
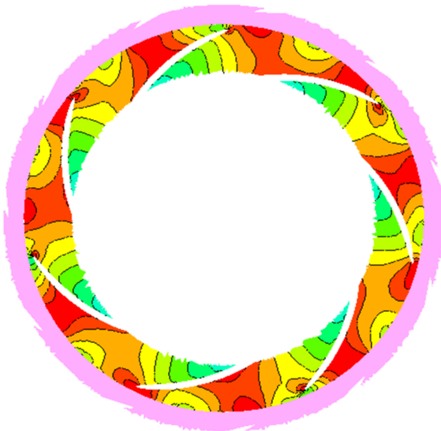
	Impeller with channel type diffuser	*Impeller with cascade type diffuser*
Scale	C_r^* 	C_u^* 
C_r^*		
C_u^*		

Figure 7 : Normalized Radial and Tangential Velocity Contour Plots at Impeller Outlet

4.2 Radial and tangential velocity chart plots at impeller outlet

Figures 8 and 9 show blade to blade circumferential chart plots of respectively normalized radial and tangential velocity at impeller mid span outlet for one blade angular position, and for both diffuser configurations. Results are taken at three constant-radius lines as defined earlier. Increasing θ is the direction of rotation.

It is quite remarkable from the figures that blade to blade velocity profiles are not similar from one impeller passage to another, predicted for both diffuser configurations. This is especially true close to the exit part of the impeller. This non periodicity of the flow structure confirms the strong interaction with diffuser vanes and with lesser effect far upstream from impeller outlet.

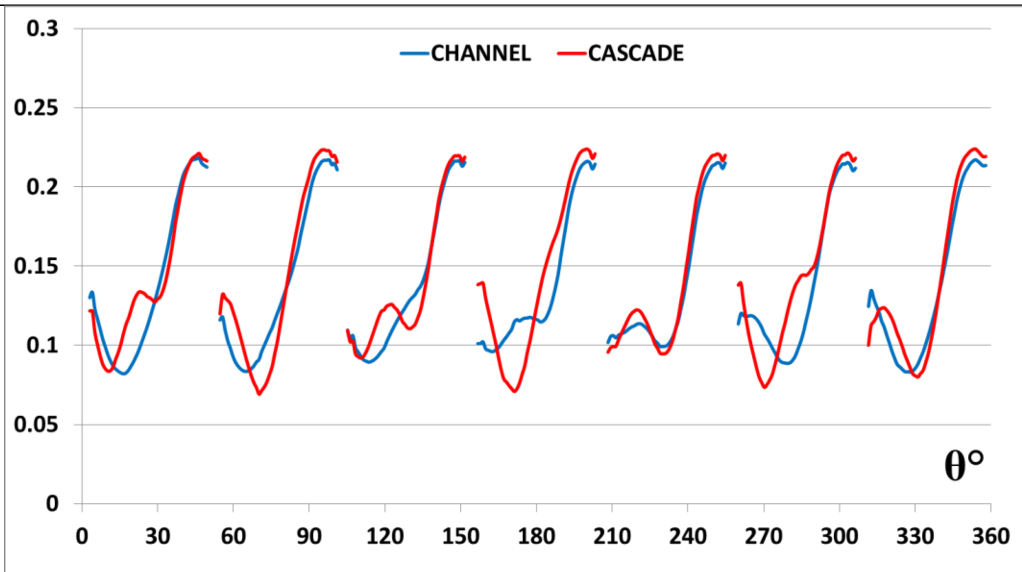
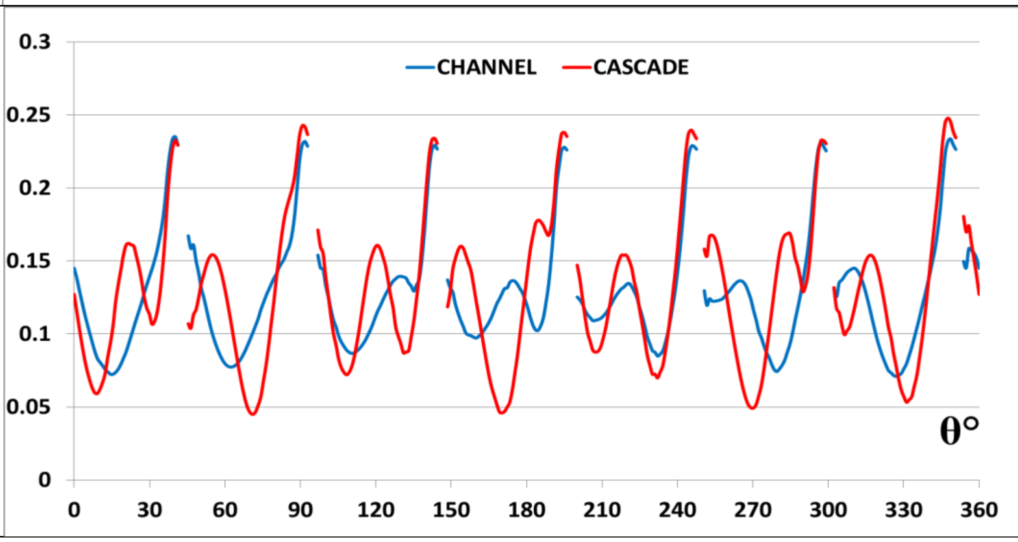
Differences do exist between velocities profiles issued for each radius, that is:

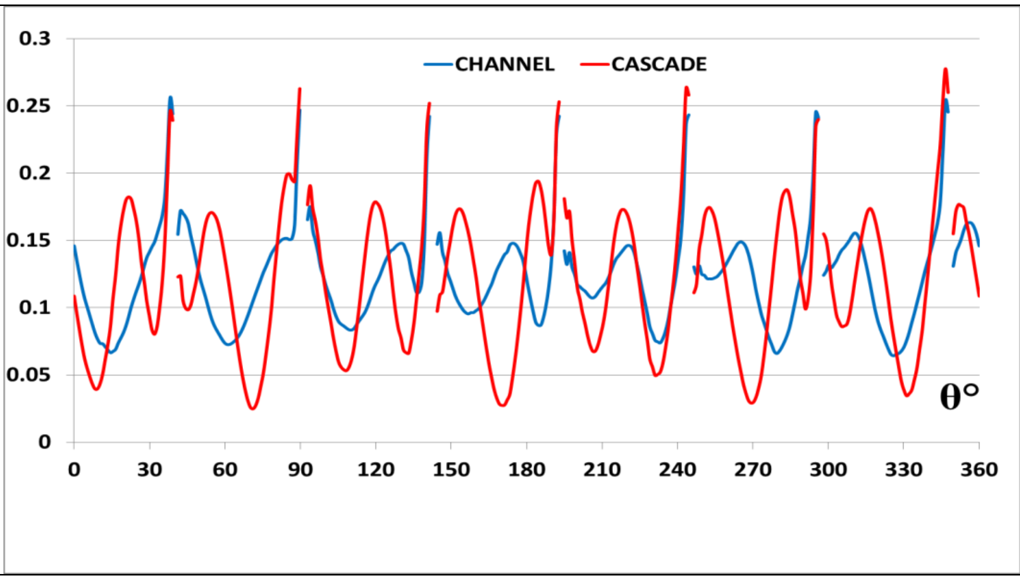
For R_1^* (Figure 8 and 9), both velocity profiles show a similar and close trend from blade to blade. A slight difference do exit, from pressure side to mid passage. At this stage, the effect of interaction is least.

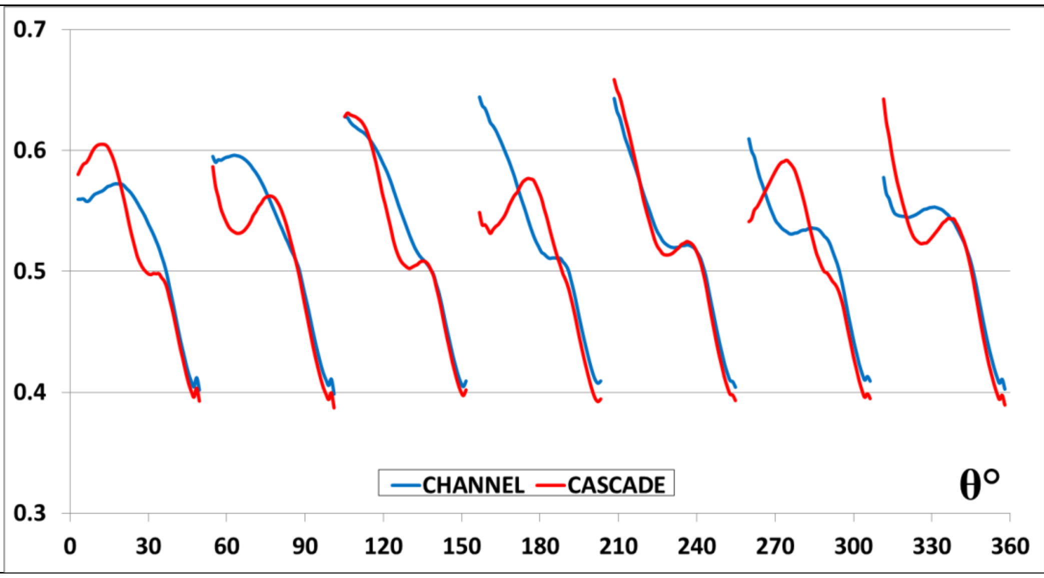
For R_2^* (Figure 8 and 9), both velocity profiles are close near the suction side of the blade, while from pressure side to mid passage, predictions give more distorted profiles in case of cascade type diffuser than for channel type. Numerous vanes have a tendency to magnify the effects of interaction within the impeller.

For R_3^* (Figure 8 and 9), both velocity profiles are different in trend and magnitude over the whole passage, except very close to suction side. Predictions give rather higher distorted profiles, with peak values corresponding to the number of vanes, in case of cascade type diffuser more than for channel type. The effects of interaction are intense. This is further true since this zone is strongly exposed due to the proximity of diffuser vanes.

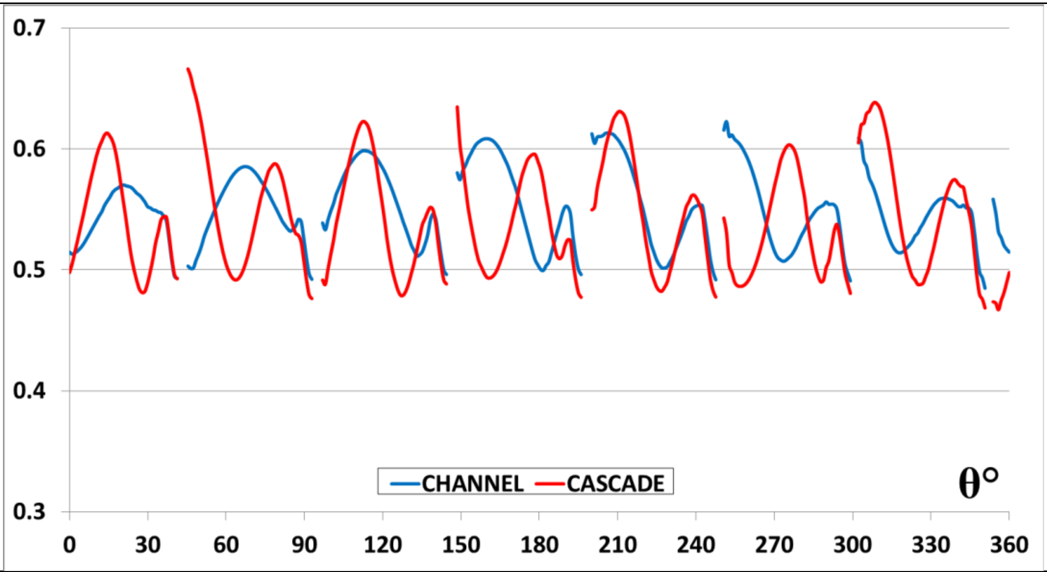
It can be seen from the figures, that interaction effects is mostly reflected on the flow in the pressure side of the blades, that is the impeller pressure side is more exposed to diffuser vanes influence as reported by other authors [4].

<table>
<tr><td></td><td>C_r^*</td></tr>
<tr><td>R_1^*</td><td>
</td></tr>
<tr><td>R_2^*</td><td>
</td></tr>
</table>

R_3^*	
	Figure 8 : Normalized Radial Velocity Profile Plots at Impeller Outlet for three radii

	C_u^*
R_1^*	

R_2^*



R_3^*

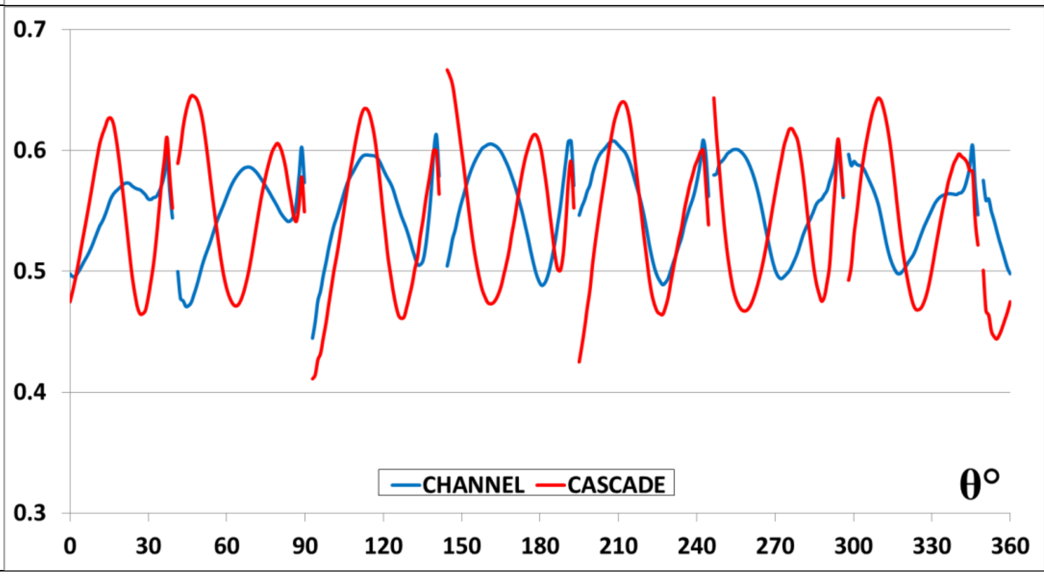


Figure 9 : Normalized Tangential Velocity profile Plots at Impeller Outlet for three radii

4.3 Numerical/Experimental Comparison of Velocity profiles

PIV measurements are available for both vaned diffuser configurations. Figure 10 shows comparison between computed results and experimental measurements for channel type diffuser only. Due to laser light reflection problems on the blades, PIV measurements for cascade type diffuser are of poor quality to be presented.
The data are extracted for the same three radii defined earlier. The velocity profiles are in relatively good agreement, since slopes and magnitudes inside the impeller are well predicted for both velocity components.

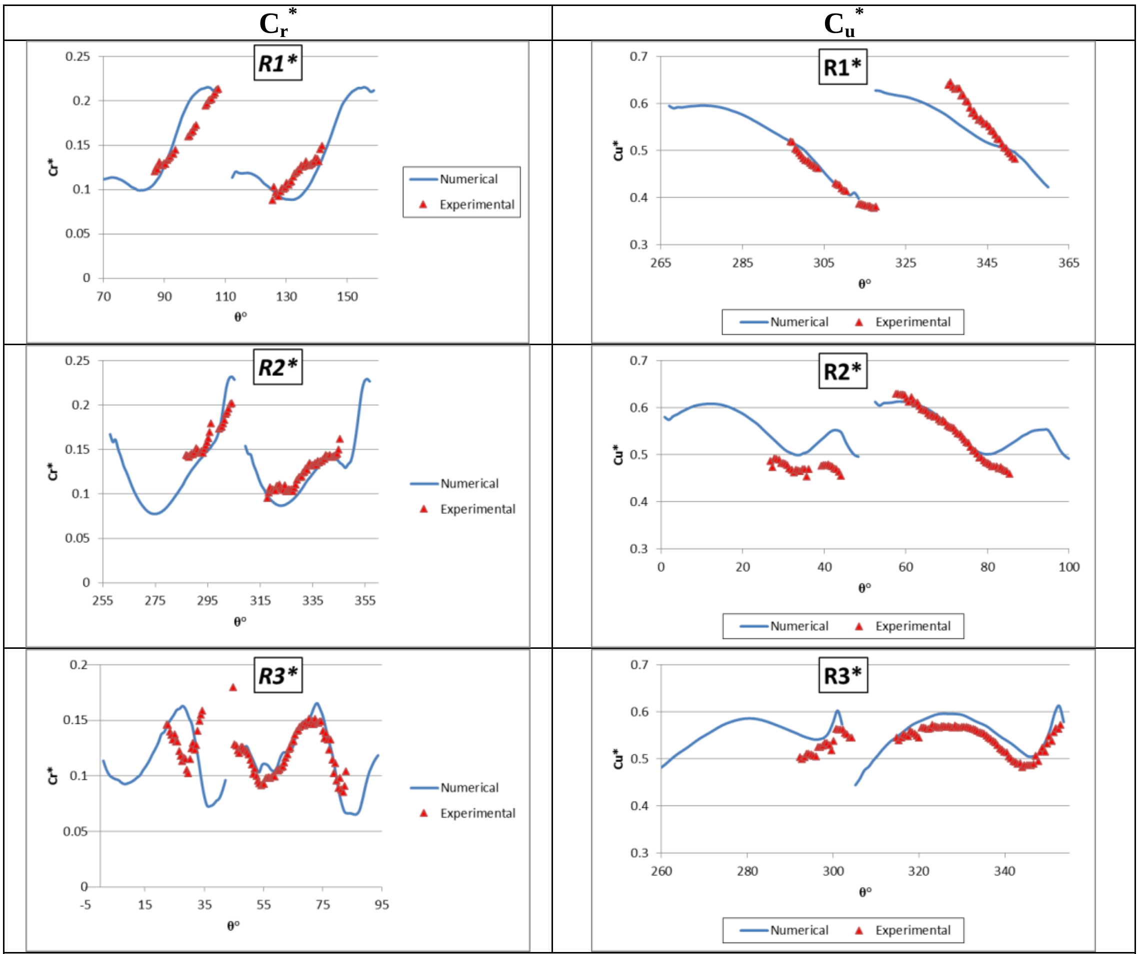


Figure 10 : Numerical/Experimental Comparison of Velocity profile Plots at Impeller Outlet with channel type diffuser

5 Conclusion

An analysis of the full 360° centrifugal pump impeller outlet flows with two vaned diffuser configurations by unsteady numerical simulations have been presented and compared. It appears clearly from the results that the character of the flow is strongly influenced by this rotor-stator interaction, even far upstream from impeller outlet. Cascade type vaned diffuser causes a more distorted flow at impeller outlet than does the channel type. Numerical predictions fit well experimental PIV measurements. We should not conclude on global pump performances. A thorough flow analysis and exploration at other flow rates would clear up the impact of cascade type or channel type vaned diffusers.

NOMENCLATURE

C_r	absolute radial velocity	[m/s]
C_u	absolute tangential velocity	[m/s]
C_r^*	normalized absolute radial velocity (C_r/U_o)	[no units]

C_u^*	normalized absolute tangential velocity (C_u/U_o)	[no units]
Q	flow rate	[m^3/s]
Q_n	design flow rate	[m^3/s]
Q^*	normalized flow rate (Q/Q_n)	[no units]
R	radius	[m]
R_o	impeller outlet radius	[m]
R^*	normalized radius (R/R_o)	[no units]
U_o	impeller outlet peripheral velocity ($\Omega\ R_o$)	[m/s]
Ω	angular velocity	[rad/s]
φ	blade angular location	[°]
θ	circumferential location angle	[°]

References

[1] Atif A., Benmansour S., Bois G., Numerical Investigation of Velocity Flow Field inside an Impeller Air Model of a Centrifugal Pump with Vaned Diffuser Interactions and Comparison with PIV Measurements, Int. J. of Rotating Machinery (2010) 1-9.

[2] Cavazzini G., Dupont P., Pavesi G., Dazin A., Bois G., Atif A., Cherdieu P., 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, Hamamatsu, Japan, 2011.

[3] Cavazzini G., Dupont P., Dazin A., Pavesi G., Bayeul A.C., Bois G., Unsteady Velocity PIV Measurements and 3D Numerical Calculation Comparisons Inside the Impeller of a Radial Pump Model, in Proceedings of the 10th European Conference on Turbomachinery, Fluid Dynamics and Thermodynamics, Lappeenranta, Finland, 2013.

[4] Feng J., Benra F.K., Dohmen H.J., “A Study on Impeller-Diffuser Interactions in a Radial Pump”, Proceedings of the 7th European Conference on Turbomachinery, Fluid Dynamics and Thermodynamics, Athens, Greece, 2007.

[5] Pavesi G., Cavazzini G., Dupont P., Coudert S., Caignaert G., Bois G., Analysis of Rotor-Stator Interactions Effects within the Vaned Diffuser of a Radial Flow Pump, Proceedings of the 23rd IAHR Symposium on Hydraulic Machinery and Systems, Yokohama, Japan, 2006.

[6] Sinha M., Katz J., Quantitative Visualization of the Flow in a Centrifugal Pump with Diffuser Vanes. Part I: On Flow Structures and Turbulence, J. Fluids Engineering 122 (2000) 97–107.

[7] Wuibaut G., Etude par vélocimétrie par images de particules des interactions roue-diffuseur dans une pompe centrifuge, Doctorate thesis, ENSAM Lille, France, 2001.

[8] Wuibaut G., Bois G., Caignaert G., Dupont P., Stanislas M., Experimental Analysis of Interactions between the Impeller and the Vaned Diffuser of a Radial Flow Pump, Proceedings of the 21st IAHR Symposium on Hydraulic Machinery and Systems, Lausanne, Switzerland , 2002.

[9] Wuibaut G., Dupont P., Caignaert G., Bois G., Rotor Stator Interactions in a Vaned Diffuser Radial Flow Pump, Proceedings of the 22nd IAHR Symposium on Hydraulic Machinery and Systems, Stockholm, Sweden, 2004.

[10] Wuibaut G., Bois G., El Hajem M., Akhras A., Champagne J Y.: “ Optical PIV and LDV comparisons of Internal Flow Investigations in SHF Impeller, Int. J. of Rotating Machinery (2006) 1-9.