International Journal of Fluid Machinery and Systems
Vol. 8, No. 3, July-September 2015

DOI: 10.5293/IJFMS.2015.8.3.193
ISSN (Online): 1882-9554

Original Paper

Leakage Flow Influence on SHF pump model performances

Patrick Dupont[1], Annie-Claude Bayeul-Lainé[2], Antoine Dazin[2], Gérard Bois[2], Olivier Roussette[2] and Qiaorui Si[3]

[1] LML, UMR CNRS 8107, Ecole Centrale de Lille, Cité Scientifique - CS20048, 59651 Villeneuve d'Ascq Cedex, France, Patrick.dupont@ecl-lille.fr
[2] LML, UMR, CNRS 8107, Arts et Métiers PARISTECH, 8, boulevard Louis XIV 59046 Lille Cedex, France, annie-claude.bayeul-laine@ensam.eu, antoine.dazin@ensam.eu, Gerard.bois@ensam.eu, olivier.roussette@ensam.eu
[3] Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, Zhenjiang, Jiangsu 212013, China, siqiaorui@163.com

Abstract

This paper deals with the influence of leakage flow existing in SHF pump model on the analysis of internal flow behaviour inside the vane diffuser of the pump model performance using both experiments and calculations. PIV measurements have been performed at different hub to shroud planes inside one diffuser channel passage for a given speed of rotation and various flow rates. For each operating condition, the PIV measurements have been trigged with different angular impeller positions. The performances and the static pressure rise of the diffuser were also measured using a three-hole probe. The numerical simulations were carried out with Star CCM+ 9.06 code (RANS frozen and unsteady calculations). Some results were already presented at the XXth IAHR Symposium for three flowrates for RANS frozen and URANS calculations. In the present paper, comparisons between URANS calculations with and without leakages and experimental results are presented and discussed for these flow rates. The performances of the diffuser obtained by numerical calculations are compared to those obtained by the three-holes probe measurements. The comparisons show the influence of fluid leakages on global performances and a real improvement concerning the efficiency of the diffuser, the pump and the velocity distributions. These results show that leakage is an important parameter that has to be taken into account in order to make improved comparisons between numerical approaches and experiments in such a specific model set up.

Keywords: pump, numerical calculations, performances, unsteady flow, vaned diffuser, radial flow pump

1. Introduction

This work is an extended study that had originated in LML laboratory concerning the well-known SHF (Société Hydrotechnique de France) pump. A lot of experimental data have already been produced (tests in air and in water) on that geometry and these results are used as databases for the validation of numerical calculation results and for new experimental results [1-8]

Flow behaviour in a radial machine is quite complex and strongly depends on rotor stator interactions and operating conditions [9-11].

In numerical simulations, two aspects have to be considered:

- The first one concerns the governing equations which are solved in the model. Three kinds of numerical calculations are currently used in turbo machinery (frozen rotor calculations, mixing plane calculations, unsteady calculation). The frozen rotor calculation is the steady state method which uses the rotating reference frame to save the computational resources by converting inherently transient turbo-machinery flow into steady state. The difference between a frozen rotor and a mixing plane calculation is that the mixing plane mixes the flow and applies the average quantities on the interface for upstream and downstream components, while a frozen rotor will pass the true flow to downstream and vice versa. So if the wake effect on the downstream component performance is necessary, the frozen rotor method has to be used. A big disadvantage is that it gives the solution at a single relative position. The true transient method gives the wake effect on the downstream component for all relative positions (as it happens in reality) [12-14].

Received January 15 2015; revised March 20 2015; accepted for publication September 2 2015: Review conducted by Prof. Soon-Wook Kim. (Paper number O15043S)
Corresponding author: Patrick Dupont, Patrick.dupont@ecl-lille.fr

This paper was presented at the 27th IAHR Symposium on Hydraulic Machinery and Systems, September 4, Montreal, Canada.

- The second aspect concerns the geometrical model. Some geometrical simplifications are currently used. For example, the leakage flows are often neglected. It is obvious that a complex model (fully unsteady, with leakage flows) will be more time consuming but closer to the real physics. In the presence of a gap, a leakage flow appears due to the difference of static pressure at the clearance between the shroud and the outer casing and between the hub and the outer casing.

In this respect, the effects of fluid leakage due to the gap between the rotating and the fixed part of this pump model are analysed and discussed.

In this paper, some limits of the numerical models are pointed out. To do so, several numerical calculations have been carried out: i-Frozen rotor without leakage, ii-Unsteady RANS calculations without leakage, iii-Frozen rotor with leakage and iv-Unsteady RANS calculations with leakage

These numerical results are compared to experimental results for three flow rates to try to evaluate the effects of the numerical models on the prediction of the performance and on the local behaviour of the flow.

2. Experiments

2.1 Test and apparatus

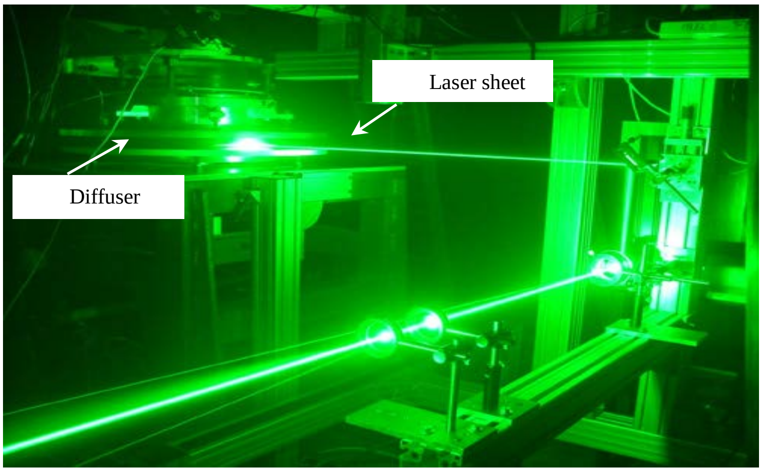


Fig. 1. Apparatus: PIV measurement, laser sheet.

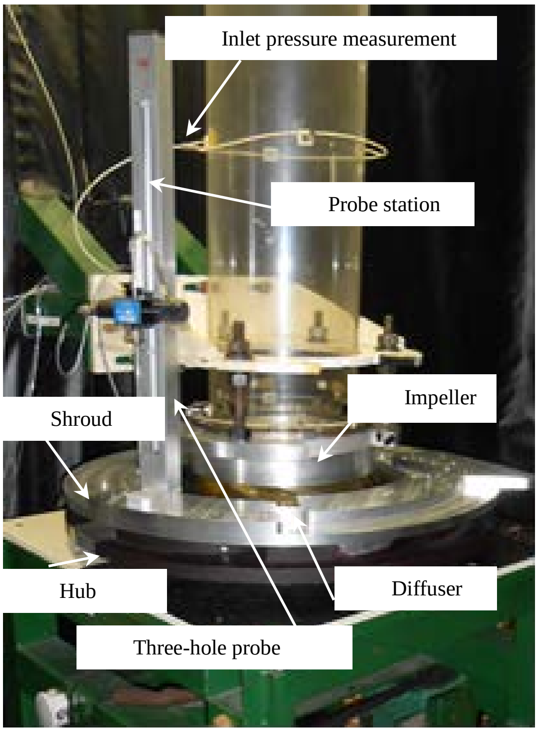


Fig. 2. Apparatus: three-hole probe.

The test model corresponds to the so-called SHF pump, working with air, in similarity conditions compared to water, for which several studies have been made [1-5;15-16] involving numerical and PIV comparisons. The Optical assembly to create a laser sheet in one diffuser blade passage can be shown in figure 1.The existing database has been completed by pressure probe measurements for a complete performance analysis in the vane diffuser part of the pump model. The test rig used for the three-hole probe (figure 2) was the same as the one already developed for the PIV measurement already described in the previous papers, especially in reference [4].

Table 1. Pump characteristics

Impeller		**Diffuser**	
Inlet radius	$R_1 = 0.14113$ m	Inlet radius	$R_3 = 0.2736$ m
Outlet radius	$R_2 = 0.2566$ m	Outlet radius	$R_4 = 0.3978$ m
Number of blades	$Z_i = 7$	Number of vanes	$Z_d = 8$
Outlet height	$b_2 = 0.0384$ m	Height	$b_3 = 0.04$ m
Impeller design flow rate	$Q_i = 0.337 m^3/s$	Diffuser design flow rate	$Q_d = 0.8Q_i$
Rotational speed	N = 1710 rpm	Impeller-Diffuser radial gap	1 mm

.

Test pump model and PIV measurements conditions have been already described in several papers [3-4] and main pump characteristics are given in table 1.

Two sources of fluid leakage occur:

- The first one at the impeller inlet (leakage 1, figure 6)
- The second one between the impeller outlet and the diffuser both for the hub and the shroud sides (leakages 2 and 3, figure 6).

2.2 PIV measurements

PIV measurements have been performed at different hub to shroud planes inside one diffuser channel passage for a given speed of rotation and various flow rates. For each operating condition, the PIV measurements have been trigged with different angular impeller positions (figure 3). For each angular position, four hundred instantaneous velocities charts have been obtained, covering the space between inlet and outlet diffuser throats. This makes a rather good evaluation of phase averaged velocity charts possible. The PIV results are extracted from the thesis of G. Cavazzini, [2]. In order to do comparisons with the other methods, for each probe position (see paragraph 2.3), mean values of radial, tangential and absolute velocity angle α relative to radial direction inside the diffuser were calculated for the amount of the four hundred instantaneous and of the nine each angular positions.

2.3 Three holes probes

A directional three holes probe has been used to make hub to shroud traverses [17]. Using a specific calibration one can get total pressure, static pressure, absolute velocity and its two components in radial and tangential direction.

In order to well represent the flow field, twenty-three probe locations are defined as it can be seen in figure 5. For each location, ten axial positions are registered (b^*=0.125, 0.2, 0.25, 0.375, 0.5, 0.625, 0.75, 0.875, 0.925, 0.975 from hub to shroud). The present analysis focuses only on locations 19 to 23 along the blade to blade diffuser channel.

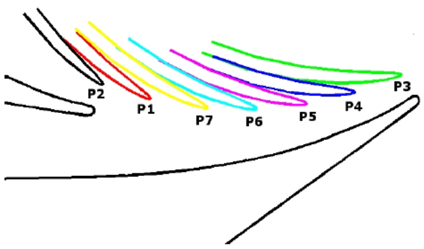


Fig. 3. Impeller different angular positions relative to the diffuser vanes

Fig. 4. Probes for frozen rotor calculations.

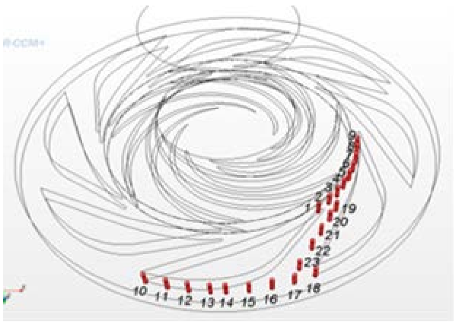
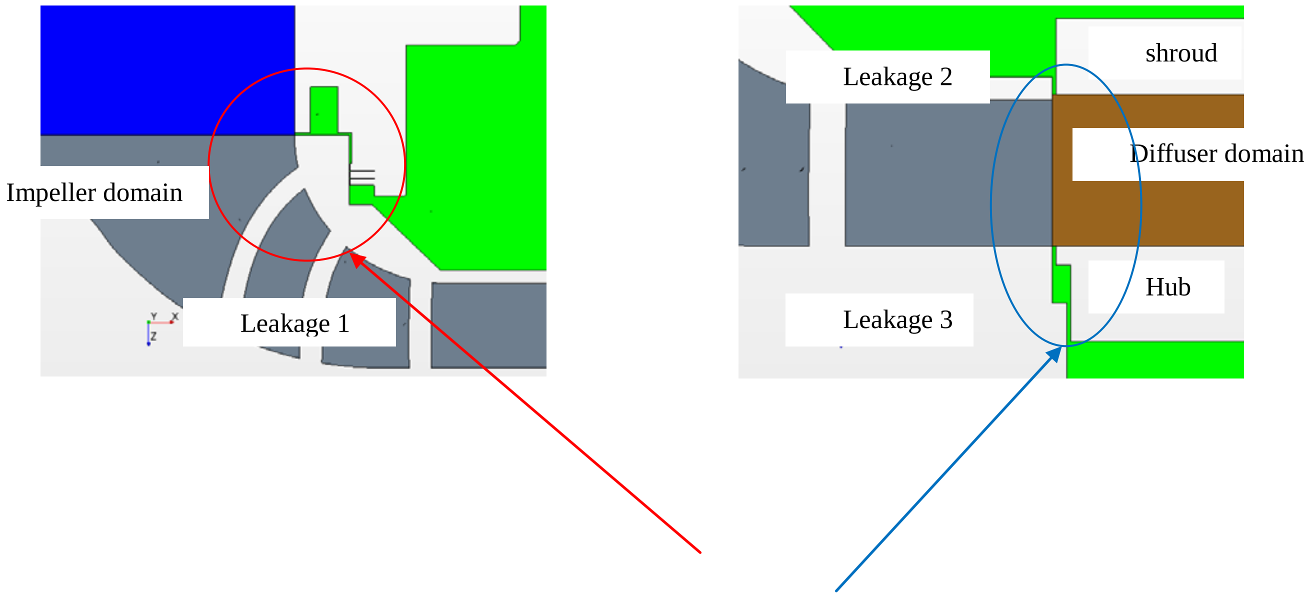


Fig. 5. Diffuser measurement locations for probe traverse and unsteady calculations.

3. Calculations



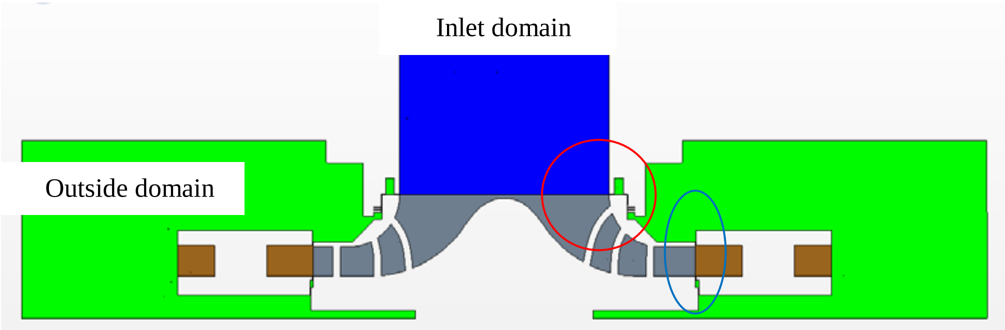


Fig. 6. Regions modelled. Details of fluid leakages.

Calculations were carried out with Star CCM+ 9.06 code. The sketches of the gaps are really taken account (figure 6) in the model with leakages. A complete meshing was set up with the real geometry of the gaps and external domain far upstream the labyrinth. In this respect, boundary conditions include the relative casing motion of the model.

The calculation domain is divided into four domains:

- Inlet domain,
- Impeller domain,
- Diffuser domain,
- Exterior domain as can be seen in Figure 6.

As thickness gaps are very small (1 mm), a thin meshing model has been chosen. A thin meshing model allows thin regions in the geometry to have a prismatic type volume mesh. Using this kind of mesh improves the overall cell quality and reduces the cell amount when compared to an equivalent polyhedral type core mesh. The number of layers in the thin mesh has been set to 10. In the other parts of the domain, a polyhedral mesh with prism layers is used for all calculations (5 prism layers for a total prism layer thickness of 1 mm). The target size is 3 mm and the minimum size 0,5 mm . The size of the grid is about 20. 10^6 cells for all the four domains. Concerning the model “without leakages, only Inlet, Impeller and diffuser domains are taken account. The final grid is about 10. 10^6 cells with the same parameters (base size, surface size, number and size of prism layers).

Three-dimensional incompressible Reynolds averaged Navier–Stokes equations are solved in steady (frozen rotor) and unsteady states. The SST k-ω turbulence model is used [7, 18].

The boundary condition at the inlet consisted of a mass flow rate (Q*=0.766; 0.973; 1.134). The boundary condition at the outlet was the atmospheric pressure (relative pressure=0 Pa). The boundaries of the outer casing of the impeller are considered as rotating walls. The fluid (air) was considered incompressible at a constant temperature of 20ºC.

For frozen rotor calculations, line probes are plotted in the whole domain of the diffuser (figure 4): each probe is duplicated seven times in order to obtain all parameters (pressure, total pressure, radial, tangential and axial velocity and velocity magnitude) for different relative positions of the diffuser comparatively to the impeller (figure 3). This is equivalent to azimuthal positions equal to n* 360/(Zi*Zd) with n=0,1 ,2, 3, 4, 5, 6 and 7.

For unsteady calculations, point probes are plotted in the blade to blade channel of the diffuser as can be seen in figure 4.

The convergence criteria are less than 1.e-4. The values of y+ are below 15 in the whole computational domain.

4. Results and analysis

Let us first examine the global results: the non-dimensional static pump head, the non-dimensional isentropic pump head, the impeller efficiency, the diffuser efficiency and the pump efficiency.

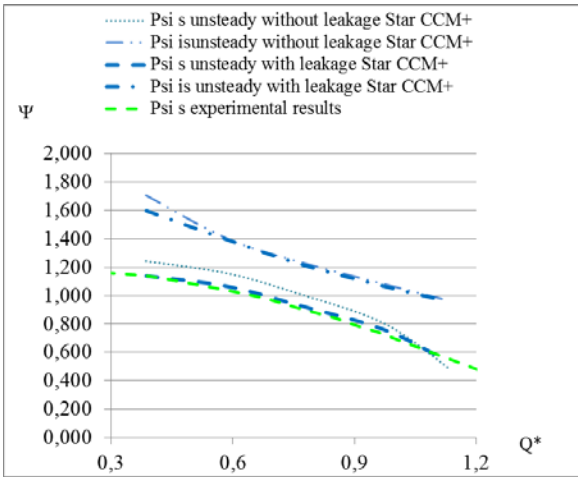


Figure 7. Performances of the pump Ψ

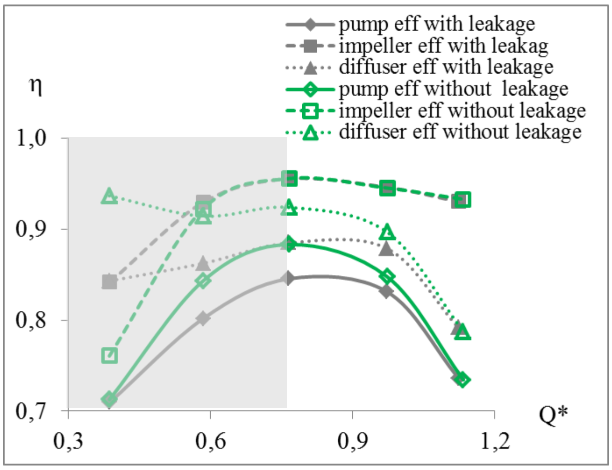


Figure 8. Impeller and pump efficiency η

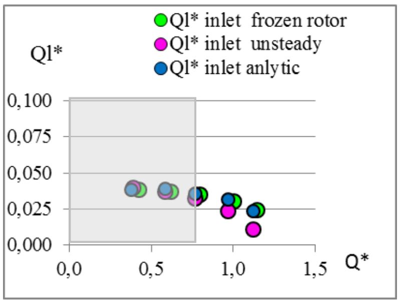


Figure 9. Fluid leakages at impeller inlet

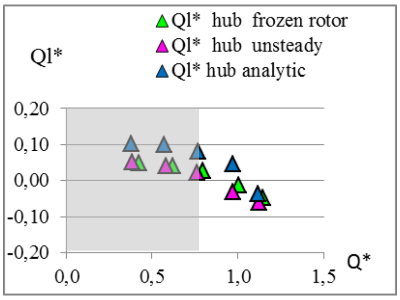


Figure 10. Fluid leakages at impeller/diffuser hub

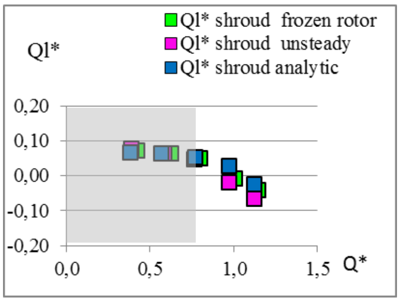


Figure 11. Fluid leakages at impeller/diffuser shroud

These results are given from Q*=0.386 to Q*=1.13, but results at low flow rate have to be more analysed concerning the efficiencies. This analysis will be given in an other paper.

These calculations show the influence of fluid leakages on global performances (figure 7). Global results of static theoretical head pump are in very good agreement between the experimental results and the unsteady calculation results with fluid leakages, as it can be seen in figure 7. The numerical non-dimensional isentropic pump head is quite the same than the experimental one when the fluid leakages are modelled in case of unsteady calculations. The numerical non-dimensional static and total head pump are smaller in case of fluid leakages than those in case of no fluid leakages. This leads to a decrease of impeller efficiency at very low flowrate due to a significant increase of the impeller moment. This result needs more analysis which is not the aim of the present paper but which can be noticed. The pump efficiency due to fluid leakages is lower than those without fluid leakages (figure 8). It can also be noticed that the pump efficiency falls at high flow rate(Q* higher than 1.1). The diffuser efficiency decreases with flow rate with and without fluid leakages. This is due to the fact that the diffuser is not well adapted at high flow rate. It is well adapted at Q*=0.766 as can be observed in figure 8.

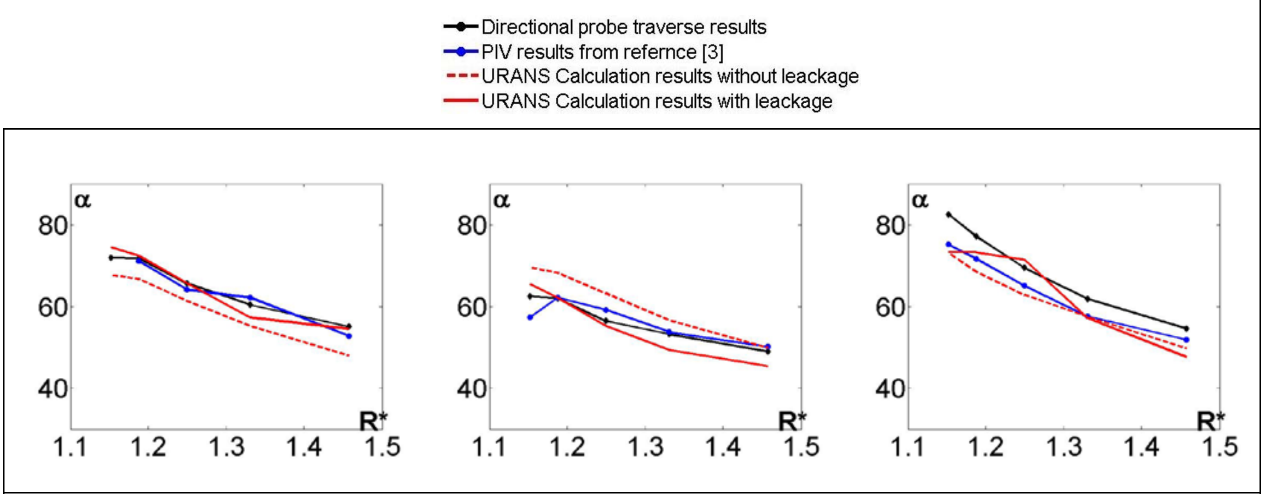


Figure 12. α for Q*=0.766

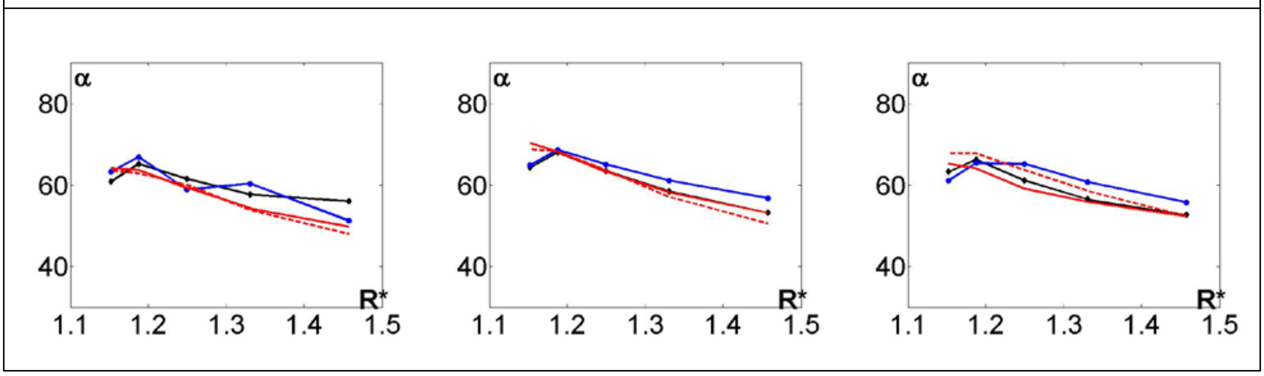
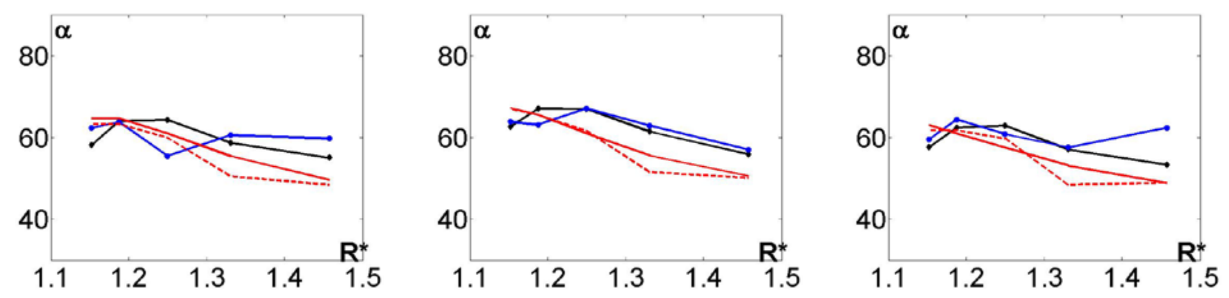


Figure 13. α for Q*=0.974

Figure 14. α for Q*=1.134



b* = 0.125	b* = 0.500	b* = 0.875

The evolution of absolute velocity angle α relative to radial direction inside the diffuser

Fluid leakages are calculated for the three different locations: inlet (leakage 1), hub side (leakage 2) and shroud side (leakage 3) with frozen rotor and unsteady calculations. The impeller rotation effect was taken into account in the numerical solution.

Wuibaut [2] and Cherdieu [17] calculated analytical solutions, based on friction losses and localized minor losses of charge, for each leakage. The impeller rotation effect was not taken into account in this analytical solution. By writing the Bernoulli's theorem, each leakage flow rate has been calculated knowing the total difference pressure between the inlet and the outlet of the gap.

Considering the leakage at the impeller inlet, only localized losses of charge in the labyrinth are taken account [2]. For the gap between the impeller and the diffuser on the shroud side, the losses are due to the contraction and the expansion through the gap for the localized part and also due to friction on the length of the ring [17]. For the gap between the impeller and the diffuser on the hub side, Cherdieu considered that the losses are the sum of two elbows, the contraction and the expansion [17].

All results are presented in Figures 9 to 11, fluid leakage are considered positive if they income inside the domain.

As can be observed, at the inlet gap, all methods lead to quite same results. In case of Star CCM+, fluid leakages are smaller at high flow rate.

Concerning, leakages at hub and shroud sides, the amount of flow rate depends on the flow rate in the pump. At low values of Q*, a positive leakage between impeller and diffuser was observed. At high values the tendency inverses as shown in figures 10 and 11.These tendencies were also observed in previous studies [7].

Considering fluid leakages flow rates, the analytical results seem to be sufficient enough to predict these flow rates, unless at the hub side. The prediction of such flow rates was also shown by Khelladi et al [19].

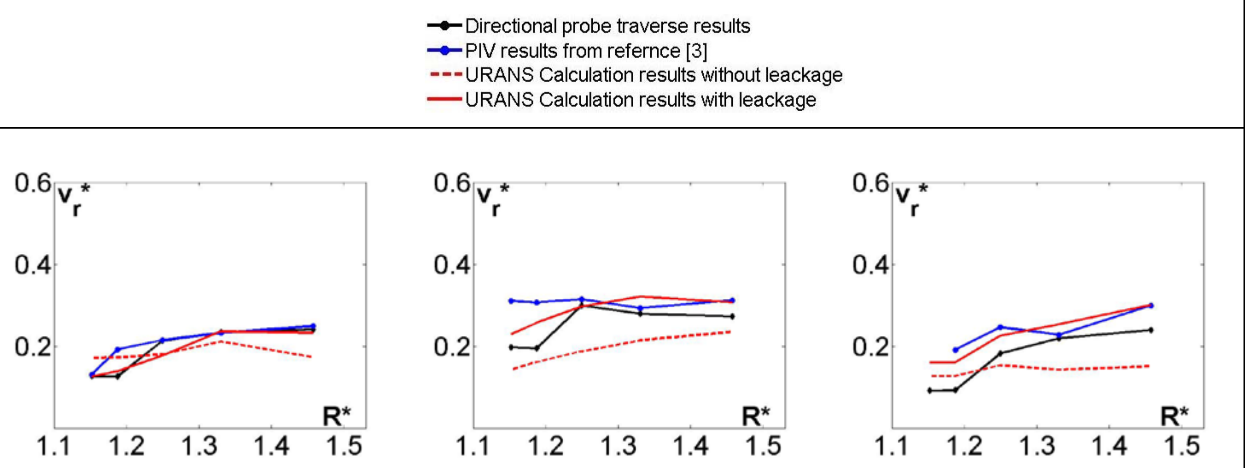


Figure 15. V_r* for Q*=0.766

Figure 16. V_r* for Q*=0.974

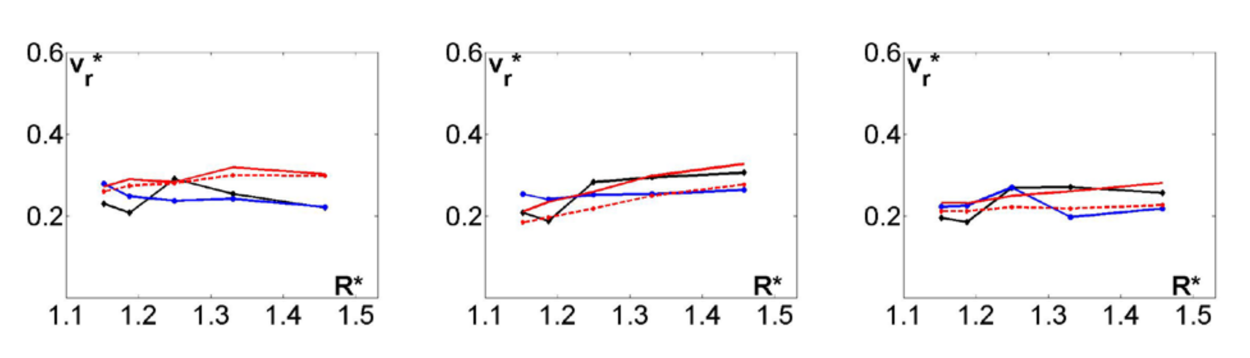
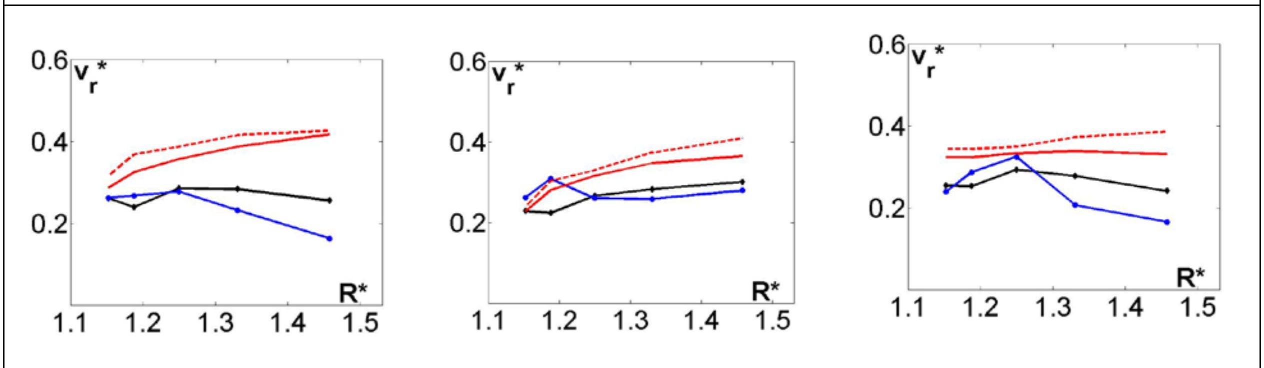


Figure 17. V_r* for Q*=1.134



b* = 0.125	b* = 0.500	b* = 0.875

The evolution of non-dimensional radial velocity V_r*inside the diffuser

Local absolute velocity angle relative to radial direction and non-dimensional radial and tangential velocity are plotted in figures 12 to 20 for the three flow rates.

The examination of the evolution of the absolute velocity angle relative to radial direction for the two numerical calculations, for the PIV measurements and for the three holes measurements show similar results (figures 12 to 14).

There is a small influence of taken account the fluid leakages on the evolution of the angles.

The examination of radial velocities (figures 15 to 17) and tangential velocities(figures 18 to 20) shows that fluid leakages have a real influence for Q*=0.776 and Q*=0.973. For Q*=1.173, numerical results with fluid leakages are in better agreement with experimental results than those without leakage, but some differences still remain with experimental results especially at low values of b*.

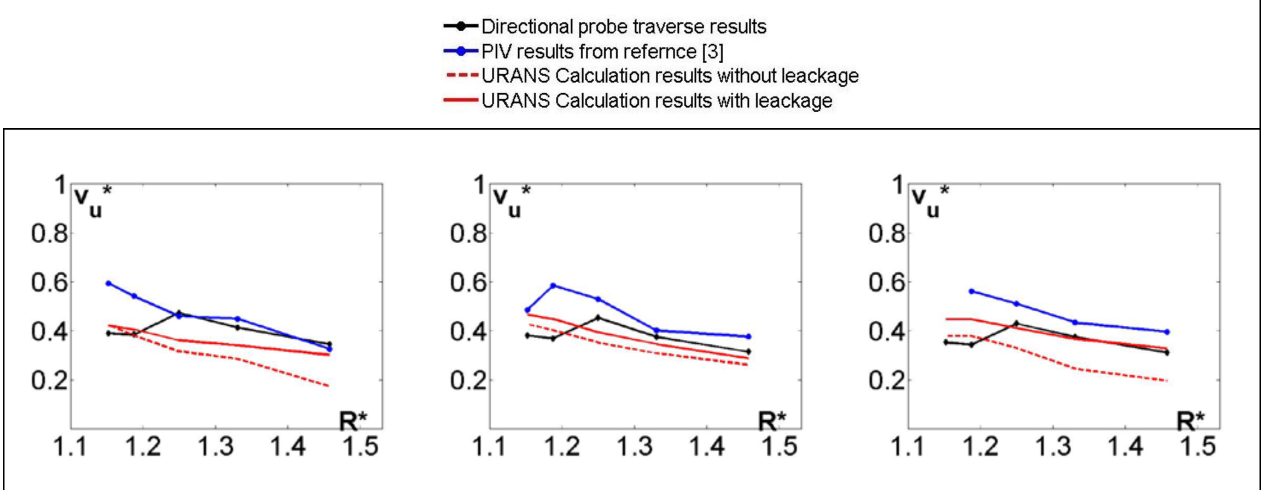


Figure 18. V_u* for Q*=0.776

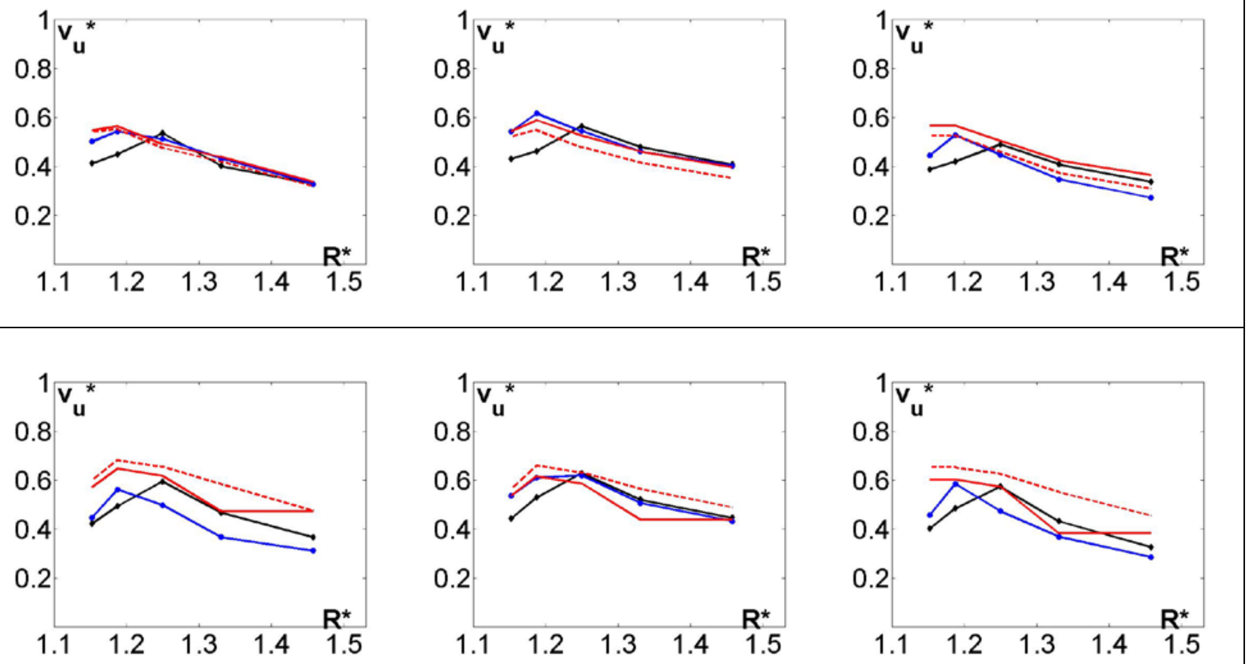


Figure 19. V_u^* for $Q^*=0.974$

Figure 20. V_u^* for $Q^*=1.134$

b* = 0.125 | b* = 0.500 | b* = 0.875

The evolution of non-dimensional radial velocity V_u^*inside the diffuser

5. Conclusion

In this paper, the SHF pump was studied numerically and experimentally. The computational models with and without gaps (between the impeller, the fixed inlet domain and the fixed diffuser domain) were set up for frozen rotor and unsteady solutions in order to determine the fluid leakages and to make comparisons with experimental results. A one dimensional loss model was also used to calculate the fluid leakages flow rates. The effects of the gaps on the calculated flow field were analysed. Local velocity components evolutions in the diffuser blade to blade channel have been calculated and analysed taking the leakage effects into account.

It has been found that the amount of fluid leakages corresponds to the one dimensional model, even if the effect of rotating wall wasn't considered in this model. This result is very interesting knowing that the model with gaps is more consuming in grid size and in computational time than the case without gaps.

It has been shown that leakages may affect the local flow characteristics depending on volume flow rates at the impeller inlet. The calculations show influence of fluid leakage on the global performance and a real improvement concerning velocity distributions. The local fields of velocities were analysed inside the blade to blade channel using the numerical results, the PIV measurements and the three holes probes. These results show that leakage is an important parameter that has to be taken into account in order to make improved comparisons between numerical approaches and experiments.

The comparison of pump performances between the calculations and the measurements showed a favourable tendency.

New analysis at low flow rates will complete the present analysis in order to explain why the behaviour inside the impeller is influenced by the fluid leakages.

Nomenclature

b	Distance from hub [m]
b^*	Non-dimensional distance from hub (=b/B)
B	Blade height [m]
C	Impeller torque [mN]
dp_s	Static difference pressure [Pa] $(=p_s-p_{s1})$
dp_t	Total difference pressure [Pa] $(=p_t-p_{t1})$
dp_{ti}	Impeller total difference pressure [Pa] $(=p_{t2'}-p_{t1})$
dp_{tp}	Pump total difference pressure [Pa] $(=p_{t4}-p_{t1})$
N	Rotational speed [rpm]
p_s	Static pressure [Pa]
p_t	Total pressure [Pa]
Q^*	Non-dimensional volume flow rate at impeller's inlet
η_p	Pump efficiency $(=\Psi_{tp}/\Psi_i)$
η_i	Impeller efficiency $(=\Psi_{ti}/\Psi_i)$
η_d	Diffuser efficiency $(=\eta_p/\eta_i)$
α	Absolute velocity angle relative to radial direction $(=Arctg(Vu/Vr))$
ρ	Air density [kg/m^3]
Ω	Angular Velocity [rad/s] $(=\pi N/30)$
Ψ_i	Non-dimensional isentropic head $(=C\Omega/(\rho Q U_2^2/2))$
Ψ_s	Non-dimensional static pump head $(=dp_s/(\rho U_2^2/2))$

	$(=Q/Q_i)$	Ψ_{tp}	Non-dimensional total impeller head $(=dp_{ti}/(\rho U_2^2/2))$
Q_d	Diffuser design flow rate [m^3/s]	Ψ_{ti}	Non-dimensional total pump head $(=dp_{tp}/(\rho U_2^2/2))$
Q_i	Impeller design flow rate [m^3/s]	***Index***	
Q_l*	non-dimensional flow rates of fluid leakage $(=Q_l/Q_i)$	d	Diffuser
R	Radius [m]	i	Impeller
$R*$	Non-dimensional radius $(=R/R_2)$	p	Pump
V_r	Radial velocity [m/s]	0	Domain inlet
V_r*	Non-dimensional radial velocity $(=V_r/U_2)$	1	Impeller inlet
V_u	Tangential velocity [m/s]	2	Blade's Impeller outlet
V_u*	Non-dimensional tangential velocity $(=V_r/U_2)$	$2'$	Impeller outlet
U_2	Frame velocity at blade's impeller outlet [m/s] $(=\Omega*R_2)$	3	Blade's diffuser inlet
Z_d	Number of diffuser blades	4	Blade's diffuser outlet
Z_i	Number of impeller blades		

References

[1] Wuibaut G., Dupont P., Caignaert G., Stanislas M., 2000, "Experimental analysis of velocities in the outlet part of a radial flow pump impeller and the vanelessdiffuser using particle image velocimetry", Proceedings of the XX IAHR Symposium(Charlotte USA) 6-9 august, paper GU-03.

[2] Wuibaut G., 2001, "Etude par vélocimétrie par images de particules des interactions roue-diffuseur dans une pompe centrifuge", PhD thesis, Thèse de Doctorat Ensam

[3] Cavazzini G., 2006, "Experimental and numerical investigation of the rotor-stator interaction in radial turbomachines", Ph.D. thesis, University of Padova, (Padova, Italy)

[4] 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", Houille blanche –Revue internationale de l'eau ,pp. 141-151

[5] Cavazzini G., Dupont P., Pavesi G., Dazin A., G.Bois, 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", Proc. of ASME-JSME-HSME Joint fluids engineering conferenceJuly 24-29,(Hamamatsu, Japan)

[6] Cherdieu P., Dupont P., Bayeul-lainé A.C., Dazin A., Bois G., 2013, "Data reduction problems using a 3 - hole directional pressure probe to investigate mean flow characteristics in the vaneless gap between impeller and diffuser radial pump", IOP Conference Series Earth and Environmental Science09/2013; 52(topic 2).

[7] Bayeul-Lainé A.C., Dupont P., Cavazzini G., Cherdieu P., Dazin A., Bois G., Roussette O., 2013, « Numerical and experimental investigations in a vaneddiffuser of SHF impeller : fluid leakage effec"t, 21 èmeCongrèsFrançais de Mécanique(Bordeaux) 26-30 août

[8] Bayeul-Lainé A.C., Dupont P. , Miccoli L., Cavazzini G., Dazin A., Pavesi G., Bois G., 2014, "Fluid leakage effect on analysis of a vaneddiffuser of SHF pump", 15th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery, ISROMAC-15, (Honolulu, Hawai, USA); 02/2014

[9] Adamczyk J. J., Celestina M. L., Chen J. P. 1994 Wake induced unsteady flows :their impact on rotor performance and wake rectification ASME International Gas Turbine and Aeroengine Congress and Exposition(The Hague, Netherlands), June 13-16, paper 94GT219.

[10] Arndt N., Acosta A.J., Brennen C.E., Caughey T.K 1990 - Experimental Investigation of Rotor – Stator Interaction in a Centrifugal Pump With Several VanedDiffusers ASME Journal of TurbomachineryVol. 112 p. 98-108.

[11] Eisele K., Zhang Z., Casey M. V., Gülich J., Schachenmann A. 1997 Flow analysis in a Pump Diffuser Part 1 : LDA and PTV Measurements of the Unsteady FlowTransactions of ASME, Journal of Fluids Engineering Vol. 119, pp. 968-977

[12] Culver R., Liu F., 2009, "Plane Method for Flutter Computation in Multi-stage Turbomachines", 47th AIAA Aerospace Sciences Meeting Including The New Horizons Forum and Aerospace Exposition January5-8, (Orlando, Florida)

[13] Petit O., Nilsson H., 2013, "Numerical Investigations of Unsteady Flow in a Centrifugal Pump with a vaned Diffuser", International Journal of Rotating Machinery, Vol. 2013, Article ID 961580, 14 pages

[14] Tamm A., Gugau M., Stoffel B,.2002, "Experimental and 3-D Numerical Analysis of the Flow Field in Turbomachines- Part I", International Congress on Quality Assesment of Numerical Simulations in Engineering (Conception – Chile)

[15] Dazin A., CoutierDelgosha O., Dupont P., Coudert S., Caignaert G., Bois G., 2008, "Rotating Instability in the VanelessDiffuser of a Radial Flow Pump", Journal of Thermal Science17, (4), pp. 368-374

[16] Dazin A., Cavazzini G., Pavesi G., Dupont P., Coudert S., Ardizzon G., Caignaert G., Bois G., 2011, "High-speed stereoscopic PIV study of rotating instabilities in a radial vanelessdiffuser", Exp Fluids, 51(1), pp. 83-93

[17] Cherdieu P., 2014, "Contrôle du décollement dans un diffuseur aubé de turbomachine centrifuge" PhD thesis,

[18] Menter, F. R. and Kuntz, M., 2002, Adaptation of Eddy Viscosity Turbulence Models to Unsteady Separated Flows Behind Vehicles in The Aerodynamics of Heavy Vehicles: Trucks, Buses and Trains Springer Asilomar CA.

[19] Khelladi S., Sarraf C, Bakir F., Rey R., 2010, "Study of a high rotational speed shrouded centrifugal, fan: aerodynamics and effects of a shroud-associated"; Proceedings of the Institution of Mechanical Engineers Part A Journal of Power and Energy, 01/2010; 224(5):691-700. DOI:10.1243/09576509JPE899