IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

Analysis of Centrifugal Pump in Diffuser Vane By Using CFD

Dr. V.R.Sivakumar[*1], Mr.Satheesh Kumar[2], Jacob Mathews[3]
[*1] Professor –Head of the Department of Mechanical Engineering,
[2] Assistant Professor –Department of Mechanical Engineering,
[3]PG scholar – Thermal Engineering -Department of Mechanical Engineering,
Rvs College of engineering and technology, Coimbatore, Tamilnadu, India

Abstract

Flow analysis in centrifugal pumps has long been an intensive subject of research. Computational Fluid Dynamics (CFD) is the present day state-of-art technique for fluid flow analysis. It was found that non-uniformities are created in different parts of the pump at off-design conditions which result in the decrease in efficiency different radial gaps. The operating characteristic curves predicted by the numerical simulation were compared with the results of model testing and are found in good agreement. The test case consists of an enshrouded centrifugal impeller with seven blades and a radial vane diffuser with 7 vanes. A large number of measurements are available in the radial gap between the impeller and the diffuse, making this case ideal for validating numerical methods. Results of steady and unsteady calculations of the flow in the pump are compared with the experimental ones, and four different turbulent models are analyzed. The steady K-£ RNG is wall defined equation function simulation uses the frozen rotor concept, while the unsteady simulation uses a fully resolved sliding grid approach. The comparisons show that the unsteady numerical results accurately predict the unsteadiness of the flow, demonstrating the validity and applicability of that methodology for unsteady incompressible turbo machinery flow computations. The steady approach is less accurate, with an unphysical advection of the impeller wakes, but accurate enough for a crude approximation. The different turbulence models predict the flow at the same level of accuracy, with slightly different results.

Keywords: Centrifugal pumps, CFD, flow analysis, numerical simulation, radial gab, part load performance.

Introduction

A centrifugal pump is a roto-dynamic machine that uses a rotating impeller to increase the pressure of a fluid. They are widely used for liquid transportation in different sectors. Their operating range spans from full-load down to close to the shut-off head. In order to develop a reliable machine for this highly demanding operation, the behavior of the flow in the entire pump has to be predicted before they are put in actual use.

This requires critical analysis of highly complex flow in the pump which is turbulent and three dimensional in nature. The flow analysis through experiments or model testing is considered to be time consuming, tedious and expensive. CFD is the present day state-of-art technique in fluid flow analysis. In recent years, most of the industries are using CFD as a numerical simulation tool for flow analysis of centrifugal pumps. Due to the development of CFD code, one can predict the efficiency of the system as well as observe actual behavior. One can find the root cause for poor performance by using CFD analysis of the system.

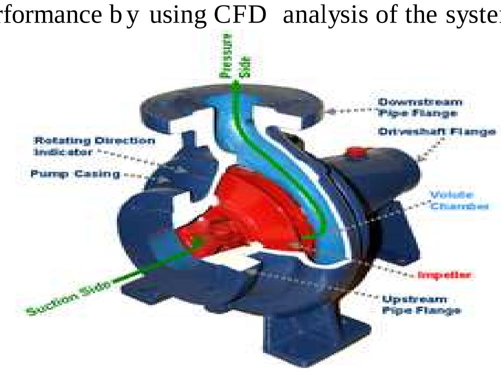

Figure 1.1 centrifugal pump with diffuser vane

Many researchers have used CFD for the numerical simulation of centrifugal pumps. Patel and Ramakrishnan carried out CFD analysis of mixed flow pump (Nsq. =46.00 Metric) at duty point and also at part load conditions.

ISSN: 2277-9655
Impact Factor: 1.852

Muggli et al., 1997 applied Navier-Stokes code with the standard k- turbulence model for CFD analysis of highly loaded pump diffuser flows. Hamkins and Bross, 2002 have shown how modern image analysis methods allow quantitative predictions of the corresponding pressure distribution by analyzing surface flow patterns. They mentioned that the surface flow patterns can also be used to adjust boundary conditions for CFD simulations by trial and error until a good match with the measured pattern can be found.

Medvitz et al., 2002 used multi-phase CFD method to analyze centrifugal pump performance under cavitating conditions. They used homogeneous two phase Reynolds-Averaged- Navier-Stokes equations, wherein mixture momentum and volume continuity equations were solved along with vapour volume fraction. Zhou et al., 2003 carried out numerical simulation of internal flow in three different types of centrifugal pumps (one pump has four straight blades and the other two have six twisted blades). A commercial three-dimensional Navier-Stokes code called CFX, with a standard k- two- equation turbulence model was used. They found that the predicted results relating to twisted-blade pumps were better than those relating to the straight-blade pump, which suggests that the efficiency of a twisted-blade pump will be greater than that of a straight-blade pump.

Bacharoudis et al., 2008 carried out parametric study of impellers with the same outlet diameter having different outlet blade angles. The numerical solution of the discretized three-dimensional, incompressible Navier- Stokes equations over an unstructured grid was accomplished with a commercial CFD code.

Spence and Teixeira, 2009 used multi-block, structured grid CFD code to carry out parametric study of double entry, double volute centrifugal pump. The cutwater gap and vane arrangement were found to exert greatest influence across various monitored locations and flow range.

1. In this paper, flow analysis of 200 m^3/hr capacity centrifugal pump carried out using commercial CFD package FLUENT is presented. The salient features of the centrifugal pump are given in Table 1.1

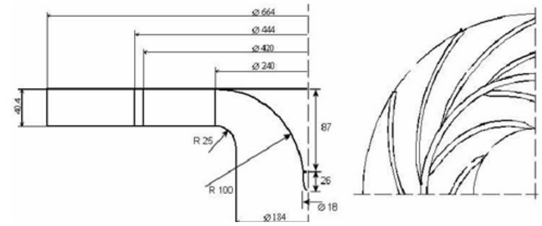

Fig 1.2 Ercoftech centrifugal pump diffuser vane Dimensions

Table 1.1 Salient features of centrifugal pump – Erocoftech model

Rated Head	20.1 m
Rated Discharge	200 m^3/ hr
Rated Speed	1440 rpm
No. of impeller vanes	7
Diameter of impeller eye	150 mm
Outer diameter of impeller	260 mm
Blade thickness	6 mm
Width of impeller at outlet	26 mm

Computational Fluid Dynamics

CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. Reduction of lead time and cost can be achieved by the use of CFD.

The flow parameters in centrifugal pumps are governed by two fundamental conservation equations namely:

(i) Mass conservation equation

(ii) Momentum conservation equation These conservation equations can be expressed in terms of partial differential equations.

CFD is a numerical technique to replace these partial differential equations of fluid flow into the algebraic equations by numbers and discrediting them in space and/or time domain. With the aid of the CFD approach, the complex internal flow in centrifugal pump which is not fully understood yet can be well predicted to speed up the pump design procedure.

Steady State Numerical Simulations

The computational domain consists of spiral casing and impeller having 7 numbers of blades. Total 12 sets of numerical steady state simulations were carried out to get the

ISSN: 2277-9655
Impact Factor: 1.852

performance data of centrifugal pump. The flow in the impeller was computed in the moving reference frame, while the flow in the casing was calculated in the stationary reference frame.

Gemetry creation

In any computational fluid dynamic analysis, the fluid domain creation plays an important role as the solution convergence depends on the mesh quality, which in turn depends on the geometry of the domain. The three dimensional model was created using bottom-up approach.

As the impeller blade profile is complex enough care was taken in creation of impeller domain in ICEM CFD which is a pre-processor of FLUENT. Casing which is relatively simple in construction was also created in ICEM CFD in a simplest way so as to get a better mesh as the quality of mesh depend on the complexity of the geometry.

Grid Generation

The grid for the three dimensional model was created in ICEM CFD. Due to the size and complexity of the pump care was taken while distribution of grid elements in the model. Considering the complexity of geometry, unstructured grid consists of triangular and tetrahedral element with ICEMCFD scheme was used. In order to capture the velocity and pressure gradients near wall a very fine structured mesh was generated using the size function option available in ICEMCFD. Between interacting components the non- conformal mesh with grid interface was created.

In order to check the influence of the grids on the results, meshes with different sizes were generated. The final mesh of casing with number of elements 376537 and for impeller with number of elements 1469037 was generated. Accordingly the total numbers of mesh elements were more than 1.8 million for the entire assembly as shown in Fig.1.1

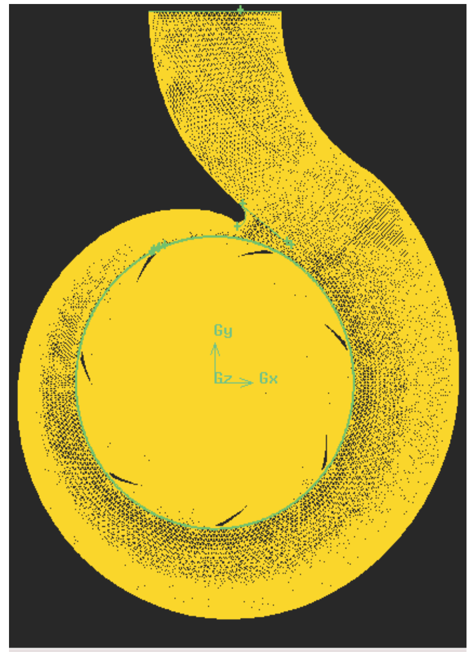

Fig. 1.3 Grid for 3-D computational model of centrifugal pump

Boundary Conditions and Turbulence Models

The simulations were carried out over a six different operating points with two different turbulence models namely renormalization group (RNG) k-model and shear stress transport (SST) k-model. Mass flow rate correspond to different operating points was specified at the suction of impeller while total pressure was defined at the casing outlet. The flow in the impeller was computed in the moving reference frame, while the flow in the casing was calculated in the stationary reference frame. Between impeller and casing grid inter face was used.

Different boundary condition for the computational domain is shown in Fig. 1.4.

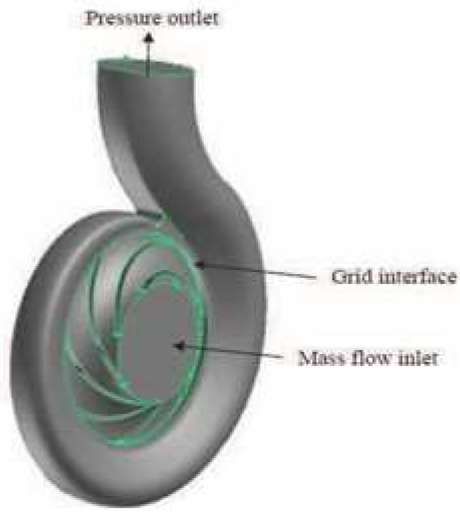

Fig.1.4. Boundary conditions for the computational domain

Solution technique

A control-volume-based technique was used to convert the governing equations into algebraic equations which were solved numerically. The flow was assumed to be steady state and water was considered as incompressible single phase fluid. The surface of all the components is assumed to be hydraulically smooth. The segregated solver with absolute velocity formulation was used. The steady state simulations were carried out at design and off-design conditions using Reynolds averaged Navier-Stokes (RANS) equations. SIMPLE scheme was used as pressure velocity coupling. And for the discretization of momentum and turbulence, second order upwind scheme were used.

Results and Discussion

To predict the centrifugal pump performance, it is essential to know how energy transfer and transformation take place in different parts of the pump. This phenomenon can be understood through the study of flow field inside the centrifugal pump. In the present study, the simulations were carried out at six different Radial gab points to points between 10% to 20% radial gab, to cover the wide range of operation, with two different turbulence models. However, only the results of CFD analysis correspond to part load (10% Radial Gab), rated (15% Radial Gab) and over rated (20% Radial Gab) conditions are discussed in the form of pressure & velocity contours, velocity vectors, path line configuration etc.

Pressure contour

Study of the static pressure contours help in understanding of energy conversion taking place in different parts of the pump. Also it is possible to locate the regions of low pressure which may be subjected to cavitation.Fig.s 3-5 shows the static pressure contours in the central plane of casing for different operating conditions. In the impeller, it is seen that the pressure continuously increases as the mechanical energy imparted in the form of impeller rotation is converted into the pressure energy. It was also observed that the pressure rise was quite gradual and uniform at rated and over rated discharge but it was non-uniform at partial discharge.

When water comes out from the impeller it contains high kinetic energy. The casing is designed such that as the water flows through the casing the kinetic energy of the water it converted into the pressure energy hence the pressure continuously increases in radial direction as well as along the flow passage.

It can be seen that the pressure variation is uniform at rated discharge but comparatively non-uniform at other operating conditions.

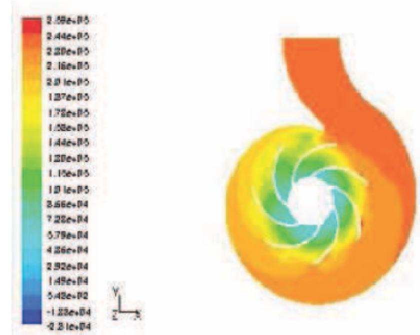

Fig. 1.5. Static pressure contour (at 10% Radial Gab-discharge values)

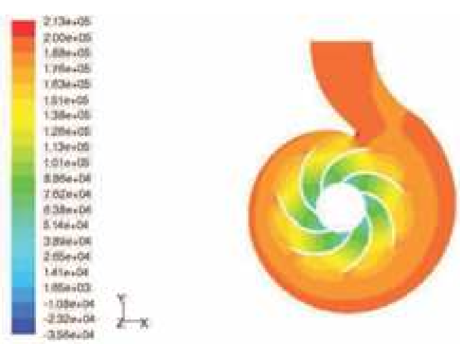

Fig.1.6. Static pressure contour (at 15% Radial Gab-discharge)

ISSN: 2277-9655
Impact Factor: 1.852

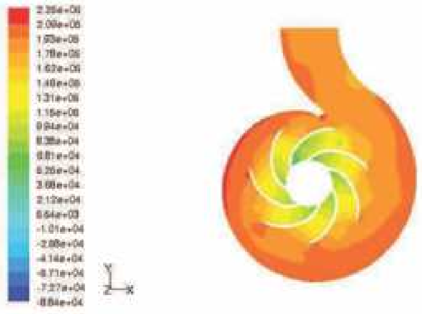

Fig.1.7. Static pressure contour (at 20% discharge)

Velocity contours

Study of the velocity contours gives idea about the kinetic energy and dynamic pressure acting in the different parts. Fig.s 5-7 shows the velocity variation in the central plane of casing for different operating conditions.

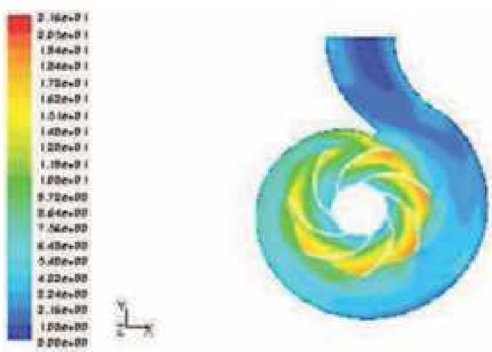

Fig.1.8. Velocity contour –common values same turbulence (at 10-20% discharge)

Mechanical energy is converted into the kinetic energy. Whereas, in the casing the velocity continuously decreases as the kinetic energy is converted into pressure energy. It can be seen that the variation of velocity is quite uniform at rated discharge compared to over load & over rated discharge.

Velocity vectors

Study of velocity vectors helps in identifying the direction of fluid particles flowing through the different components. The zones of flow separation, vortex formation and regions of secondary flow can be detected. Fig.s 9-11 shows velocity vectors at different discharge. In the impeller, the velocity continuously. Processing impeller casing

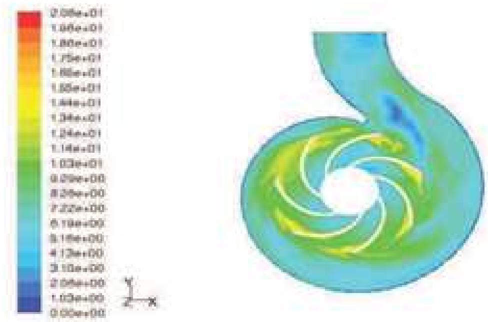

Fig.1.9. Velocity contour (at 10%Radial gab -discharge)

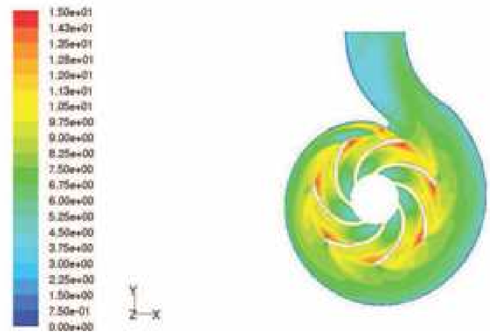

Fig.1.10. Velocity vectors (at 15% Radial Gab-discharge)

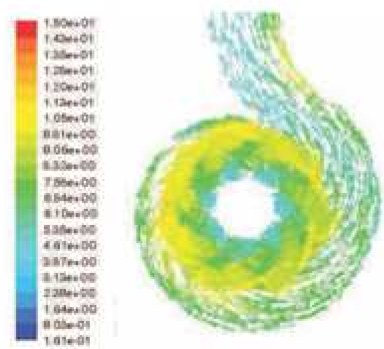

Fig. 1.11. Velocity contour (at 20% Radial Gab-discharge)

It was observed that the velocity vectors were quite uniform at rated and over rated discharge. But at partial discharge, zones of vortex formation were observed near the discharge end of the casing which may lead to energy loss and hence loss of efficiency.

Variation of parameters in radial Gabs

Fig.s 1.9 To 1.11 shows variation of velocity, static pressure and total pressure respectively in the radically outward direction towards +X axis for rated discharge. It can be seen that the velocity first increases in the impeller portion and then decreases in the casing due to the reasons as

ISSN: 2277-9655
Impact Factor: 1.852

discussed earlier. Static pressure continuously increases in radial direction. And the trend of total pressure, which is resultant of static pressure and dynamic pressure, was similar to velocity. Different turbulence of this models.

Path line configuration

Path line represents the trace or trajectory of a fluid particle over a period of time. Their study helps in understanding of particle movement identification of zones of vortex formation & secondary flow. The path lines for different operating conditions are shown in Fig.s 1.5-1.12.

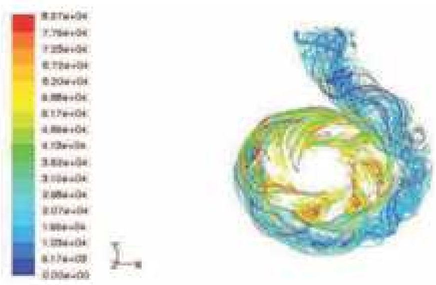

Fig.1.12. Path line configurations (at 10 % Radial Gab Total discharge)

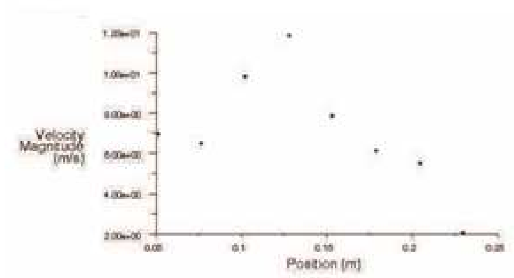

Fig 1.13 Path line configurations Graph Velocity magnitude (at Different positions)

Fig.1.12. Path line configurations (at 15% Radial Gab Total discharge)

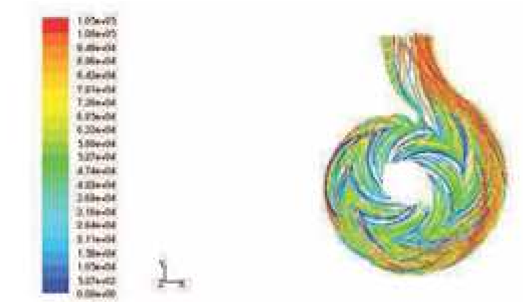

Fig.1.13. Path line configurations (at 20% Radial Gab Total discharge)

It can be seen that the path lines are non-uniform at partial discharge which result into vortex formation in the different parts of the pump. These may lead to energy loss and hence decrease in efficiency. But at rated and over rated discharge smooth path lines were observed.

Operating Characteristic Curves

For operating characteristic curves, the speed of pump was kept constant and the variation of head, power & efficiency with respect to discharge was plotted. The comparison of CFD results with the results of model tests is also presented.

Head vs discharge curve

Fig. 1.14 shows variation of centrifugal pump head with increase in discharge. As mentioned, the speed of the pump was kept constant. It can be observed that as discharge increases, head decreases. It was found that the head predicted by CFD analysis was 10 to15 % lower than the mode test results. However, the nature of head versus discharge curve is similar to that of standard pump curve.

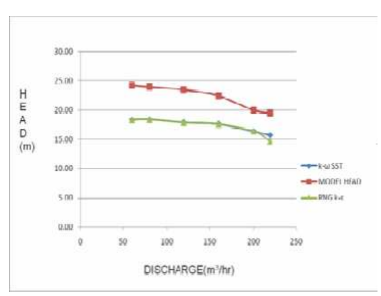

Fig. 1.14. Head versus discharge curve

ISSN: 2277-9655
Impact Factor: 1.852

Power vs discharge curve

The input power is considered as the mechanical power supplied by the motor at the pump shaft. It is calculated as the product of torque acting on the impeller and angular velocity of the impeller. The torque is considered as the resultant of pressure and viscous moments acting on the impeller (as generated by the software). The angular velocity is given as a input to the software.

In the calculations, it is assumed that the mechanical and volumetric efficiencies of the pump are 100%. Fig1.14 shows variation of input power at the pump shaft with increase in discharge. To plot this curve, the speed of the pump was kept constant and variation of shaft power input was plotted correspond to increase in discharge. It can be seen that as discharge increases the power input for the pump increases. Power predicted by CFD analysis is lower than that of model test results. However, the nature of power versus discharge curve is similar to that of standard pump curve.

Efficiency vs discharge curve

Fig. 1.15 shows variation of pump efficiency with increase in discharge. As mentioned, the speed of the pump was kept constant. It can be observed that as discharge increases, the efficiency increases, reaches maximum at rated conditions and then decreases when discharge increases beyond rated conditions, i.e. parabolic profile. It was observed that the maximum efficiency of the pump is achieved at the discharge of 200 m^3/hr as per the model testing results as well as CFD analysis.

It was found that the efficiency predicted by SST k- £ model is very close to that obtained by mode test results. But the results predicted by RNG k- £ model differ by 5% compared to model test results. However, the nature of efficiency versus discharge curve obtained by both the models was similar to that obtained by model test results.

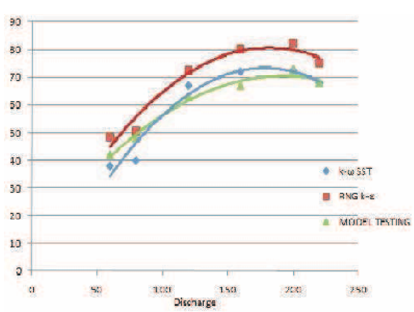

Fig. 1.15. Head versus discharge curve

Conclusion

The flow through a centrifugal pump was analyzed using commercial CFD package FLUENT for which model test results were available. The simulations were carried out at six different operating points between 10% to 20% radial gab different discharge, to cover the wide range of operation, with two different turbulence models. The following conclusions were drawn from the analysis:

- A Reynolds-averaged Navier- Stokes code with a two equation turbulence model is able to predict the important flow physics in a centrifugal pump.
- It was found that k-£- SST turbulence model provides better results compared to RNG k- £ model.
- The head predicted by CFD analysis was 10% lower Radial Gab of Discharge than the mode test results.
- The operating characteristic curves predicted by CFD were compared with the model test results & the similar trend was observed.

The findings of the present study may be useful for the analysis of centrifugal pump under similar conditions.

References

[1] Bacharoudis E.C., Filios A.E., Mentzos M.D., Margaris D.P., 2008. Parametric study of a centrifugal pump impeller by varying the outlet blade angle, The Open Mechanical Engineering Journal 2 75-83.

[2] Hamkins C.P. and Bross S., 2002. Use of surface flow visualization methods in centrifugal pump design, Journal of Fluids Engineering 124314-318.

[3] Medvitz R.B., Kunz R.F., Boger D.A., Adam J.W., Yocum A.M., Pauley, L.L., 2002. Performance analysis of cavitating flow in centrifugal pumps using multiphase CFD. Journal of Fluid Engineering 124 377-383.

[4] Muggli F.A., Eisele K., Casey M.V., Gulich J., Schachenmann A., 1997. Flow analysis in a pump diffuser-part 2: validation and limitations of cfd for diffuser flows, Journal of Fluids Engineering 119 978-984.

[5] Patel K. and Ramakrishnan N., CFD analysis of mixed flow pump.Spence R., Teixeira J.A., 2009. A CFD parametric

study of geometrical variations on the pressure pulsations and performance characteristics of a centrifugal pump. Computers & Fluids 38 1243–1257.

[6] Zhou W., Zhao Z., Lee T. S., Winoto S.H., 2003. Investigation of flow through centrifugal pump impellers using computational fluid dynamics, International Journal of Rotating Machinery 9(1) 49–61.

[7] Combès, J.F., "Test Case U3: Centrifugal Pump with a Vaned Diffuser", ERCOFTAC Seminar and Workshop on Turbomachinery Flow PredictionVII, Aussois, jan 4-7 , 1999.

[8] Ubaldi, M., Zunino, P., Barigozzi, G. and Cattanei, A., "An Experimental Investigation of Stator Induced Unsteadiness on Centrifugal ImpellerOutflow", Journal of Turbomachinery, vol.118, 41-54, 1996.

[9] Ubaldi, M., Zunino, P., Barigozzi, G. and Cattanei, A., "LDV Investigation of the Rotor-Stator Aerodynamic Interaction in a Centrifugal Turbomachine", 8th International Symposium on Applications of Laser Techniques to Fluid Mechanics, Lisbon, 1996.

[10] [4] Ubaldi, M., Zunino, P. and Cattanei, A., "Etude expérimentale de l'écoulement instationnaire dans le diffuseur aubé d'une turbomachine centrifuge", LaHouille Blanche, no 3/4, 31-37, 1998.

[11] Canepa, E., Cattenei, A., Ubaldi, M. and Zunino, P., "Wake-boundary layer interaction on the vaned diffuser of a centrifugal state", Proc. IMechE, vol. 219, Part A: J. Power and Energy, 401-411, 2005.

[12] Ubaldi, M., Zunino, P. and Ghiglione, A., "Detailed flow measurements within the impeller and the vaneless diffuser of a centrifugal turbomachine", Exp. Thermal and Fluid Sci., vol.17, 147-155, 1998.

[13] Bert, P.F., "Modélisation des écoulements instationnaires dans les turbomachines par une méthode éléments finis", Doctoral Thesis, Institut National Polytechnique de Grenoble, 1996.

[14] Bert, P.F., Combès, J.F. and Kueny, J.L., "Unsteady Flow Calculation in a Centrifugal Pump Using a Finite Element Method", Proceedings of the XVIII IAHR Symposium on Hydraulic Machinery and Cavitation, 371-380, 1996.

[15] Combès, J.F., Bert, P.F. and Kueny, J.L., "Numerical Investigation of the Rotor-Stator Interaction in a Centrifugal Pump Using a Finite Element Method", Proceedings of the 1997 ASME Fluids Engineering Division Summer Meeting, FEDSM97-3454, 1997.

[16] Torbergsen, E. and White, M.F., "Transient Simulation of Impeller/Diffuser Interactions", Proceedings of the 1997 ASME Fluids Eng Division Summer Meeting, FEDSM97-3453, 1997.

[17] Sato, K., "Blade Row Interaction in Radial Turbomachines", Ph.D. Thesis, Durham University, 1999.

[18] He, L. and Sato, K., "Numerical Solution of Incompressible Unsteady Flows in Turbomachinery", Proceedings of the 3rd ASME/JSME Joint Fluids Eng Conf, FEDSM99-6871, San Francisco, 1999.

[19] Sato, K. and He, L. "Numerical investigation into the effects of a radial gap on hydraulic turbine performance", Proc. Instn Mech Engrs, vol. 215 Part A, 99-107, 2001.

[20] Théroux, E., "Modélisation numérique des écoulements instationnaires dans les turbomachines radiales", Master Thesis, École Polytechnique de Montréal, 2003.

[21] [15] Page, M., Théroux, E. and Trépanier, J.-Y., "Unsteady rotor-stator analysis of a Francis turbine", 22nd IAHR Symposium on Hydraulic Machinery and Systems, June 29 – July 2, 2004 Stockholm - Sweden.

[22] [16] Page, M. and Beaudoin, M., "Adapting OpenFOAM for Turbomachinery Applications", Second OpenFOAM Workshop, Zagreb, 7-9 June 2007 (slides (http://powerlab.fsb.hr/ped/kturbo/OpenFOAM/WorkshopZagrebJun2007/presentations/slides/slidesPageZagreb2007.pdf))

[23] [18] Xie, S., "Studies of the ERCOFTAC Centrifugal Pump with OpenFOAM", Master's Thesis 2010:13, Chalmers University of Technology, ISSN 1652-8557, r(http://www.tfd.chalmers.se/~hani/pdf_files/ShashaMasterThesis.pdf)