

Research Article

Simulation study and Three-Dimensional Numerical Flow in a Centrifugal Pump

Lamloumi Hedi^{a*}, Kanfoudi Hatem^a, Zgolli Ridha^a^aEcole Nationale d'Ingénieurs de Tunis (ENIT. Laboratoire de Modélisation en Hydraulique et Environnement (LMHE).
B.P. 37. Le Belvédère, 1002 Tunis, Tunisia department, University1, city, countryAccepted 8 Nov.2012, Available online 1Dec. 2012, **Vol.2, No.4(Dec. 2012)**

Abstract

In this paper shows a numerical simulation of the three-dimensional fluid flow inside a centrifugal pump. For these numerical simulations the SIMPLEX algorithm is used for solving governing equations of incompressible viscous/turbulent flows through the pump. The $k-\epsilon$ turbulence model is adopted to describe the turbulent flow process. These simulations have been made with a steady calculation using the multiple reference frames (MRF) technique to take into account the impeller- volute interaction. In this study, the viscous Navier-Stokes equations are used to simulate the flow inside the vaneless impeller and volute Computational fluid dynamics (CFD) analysis is being increasingly applied in the design of centrifugal pumps. With the aid of the CFD approach, the complex internal flows in water pump impellers, which are not fully understood yet, can be well predicted, to speed up the pump design procedure. A solution method is developed to obtain three-dimensional velocity and pressure distribution within a centrifugal pump. The method is based on solving fully elliptic partial differential equations for the conservation of mass and momentum

Keywords: CFD, Centrifugal pump, 3D numerical simulation

1. Introduction

Computational fluid dynamics (CFD) analysis is being increasingly applied in the design of centrifugal pumps. With the aid of the CFD approach, the complex internal flows in water pump impellers, which are not fully understood yet, can be well predicted, to speed up the pump design procedure. Thus, CFD is an important tool for pump designers. The use of CFD tools in turbomachinery industry is quite common today. Many tasks can numerically be solved much faster and cheaper than by means of experiments. Nevertheless the highly unsteady flow in turbomachinery raises the question of the most appropriate method for modeling the rotation of the impeller.

CFD analysis is very useful for predicting pump performance at various mass-flow rate. For designers, prediction of operating characteristics curve is most important. All theoretical methods for prediction of efficiency merely give a value; but one is unable to determine the root cause for the poor performance. Due to the development of CFD code, one can get the efficiency value as well as observe actual

Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labour intensive, reducing time and, hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field

in a reasonable time. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing with variations on the simulation being performed quickly offering obvious advantages

From such literature, it was found that most previous research, especially research based on numerical approaches, had focused on the design or near-design state of pumps. Few efforts were made to study the off-design performance of pumps. Centrifugal pumps are widely used in many applications, so the pump system may be required to operate over a wide flow range in some special applications.

Thus, knowledge about off-design pump performance is a necessity. On the other hand, it was found that few researchers had compared flow and pressure fields among different types of pumps. Therefore, there is still a lot of work to be done in these fields

A centrifugal pump delivers useful energy to the fluid on pumpage largely through velocity changes that occur as this fluid flows through the impeller and the associated fixed passage ways of the pump. It is converting of mechanical energy to hydraulic energy of the handling fluid to get it to a required place or height by the centrifugal force of the impeller blade. The input power of centrifugal pump is the mechanical energy and such as electrical motor of the drive shaft driven by the prime mover or small engine. The output energy is hydraulic

* Corresponding author's Email: hedilamloumi@gmail.com

energy of the fluid being raised or carried. In a centrifugal pump, the liquid is forced by atmospheric or other pressure into a set of rotating vanes. A centrifugal pump consists of a set of rotation vanes enclosed within a housing or casing that is used to impart energy to a fluid through centrifugal force.

In the present study a three-dimensional numerical study of steady, turbulent and incompressible flow characteristics inside the passage between the blades of centrifugal pump impeller is presented. A finite volume method for solving Navier-Stokes equations in conical coordinate system with staggered grid arrangement is also presented

The flow inside the impeller of the centrifugal pump is extremely difficult to predict because of the separation and recirculation of the flow at the inlet and exit. The curvature of the blades has great influence on the flow structure.

To overcome these difficulties, some simplifications in the geometry are often considered. In the literature, many hydrodynamic models are reported in two (2D) and three dimensional (3D) by using the CFD code.

Ogut, A. et al, 2000 have provided an insight into the effectiveness of fluid injection as a boundary layer control method in suppressing or eliminating flow separation in the vaned diffuser at off-design flow conditions.

Miner, S. M et al, 2001 has worked on FLOTRAN to obtain solutions for the flow field and pressure field within the impeller of a mixed flow pump. Results produced include circumferentially averaged velocity and pressure profiles at the leading and trailing edges of the impeller. At the hub both the axial and tangential velocities were lower than the velocities at shroud and at the trailing edge the axial velocity profile shows the opposite trend from the leading edge, with the peak velocity shifted down toward the hub. Both the circumferentially averaged data and the blade passage results provide sufficient detail to evaluate the performance of the impeller.

Majidi, K. et al, 2000 have observed the secondary flow in volute and circular casings of centrifugal pumps. The static pressure was not distributed uniformly at the outlet of the impeller which results in the radial thrust. The maximum relative velocity occurred at the periphery of the impeller. The analysis shows that the curvature of the casings creates pressure gradients that cause vortices at cross-sectional planes of the casings.

(Baun, D. O. et al 2003) have observed the comparison between the characteristics of the lateral impeller forces and the hydraulic performances of four and five vane impeller operating in the spiral volute, concentric volute and double volute.

Bakir, F. et al, 2007 have observed the flow behaviour in side the impeller passage and at the intersection of the impeller and volute. The flow was relatively uniform for all the blade passages. The impeller-volute assembly requires the addition of two extended computational domains; one at the impeller inlet and the other at the volute outlet. The Frozen-Rotor interface model was considered for the study of the impeller-volute assembly. A procedure for designing the volute, the non structured

grid generation in the volute, and the interface flow passage between the impeller and volute were also discussed.

Zhou, W. et al, 2003 have used a CFD code to study three-dimensional turbulent flow through water-pump impellers during design and off-design conditions. Three different types of centrifugal pumps were considered in this simulation. One pump had four straight blades and the other two had six twisted blades. It was found that pumps having six twisted blades were better than those for pumps with straight blades, which suggests that the efficiency of pumps with twisted blades will also be higher than that of pumps with straight blades.

Hergt, P. et al, 2004 have observed the unsteady velocity, pressure and flow angle at the impeller outlet of a centrifugal pump with and without volute casing at five operating points using the hotwire technology and a fast response single hole cylindrical probe.

Anagnostopoulos, J. S. 2006 has developed a numerical model for the numerical solution of the RANS equations in the impeller of a centrifugal pump, and it was applied for a direct flow analysis and for the prediction of the impeller characteristic operation curves

Spence, R. et al, 2007 have observed that operation of centrifugal pumps can generate instabilities and pressure pulsations that may be detrimental to the integrity and performance of the pump

Younsi, M. et al, 2007 have observed the splitter blades effect on the performance of a centrifugal pump through both numerical simulation and experimental results

Cheah, K. W. et al, 2007 have numerically calculated the centrifugal pump internal flow field by using numerical methods and compared with experimental data over the wide flow range. The numerical simulation has permitted to study the internal flow pattern and pressure distribution of the pump

2.Theoretical formulation

2.1 Governing equations

Casting of The three-dimensional and incompressible flow in the pumps can be described with the conservations laws of movement and mass in cylindrical coordinates for radial (r), angular (θ) and axial (z) directions. In terms of the divergence theorem, the continuity equation or conservation mass,

Continuity:

$$\nabla \cdot \vec{V} = 0 \quad (1)$$

Momentum:

$$\rho[\vec{V} \cdot (\nabla \cdot \vec{V})] = -\nabla P + \mu_{ef}(\nabla^2 \cdot \vec{V}) + \vec{F} \quad (2)$$

where: ρ = density fluid \vec{V} = vector velocity
absolute μ_{ef} =viscosity effective fluid

P = pressure body forces

F_i : the additional sources of momentum

Since the momentum equations are considered in a relative reference frame associated to the rotor blade, the Coriolis

force and centrifugal forces are added as a momentum source term:

$$F_i = F_{i,co} + F_{i,ce} \quad (3)$$

where:

$$F_{i,co} = -2\varepsilon_{ijk}\omega_j u_k \quad (4)$$

$$F_{i,ce} = -\omega_j \omega_i x_j + \omega_j \omega_j x_i \quad (5)$$

ω_i : is angular velocity

ε_{ijk} is Levi-Civita third order tensor

2.2 Turbulence model

The left side term in Equation (2) represents the convective acceleration. The right side terms represent the pressure gradient, the viscous effects and the source terms respectively. The turbulence model chosen was the k-ε model due to its stability, widespread application in commercial softwares and robustness. The k-ε model and its extensions resolve the partial differential equations for turbulent kinetic energy k and the dissipation rate ε as shown by the Equations (6) and (7):

$$\rho[\vec{V} \cdot (\nabla.k)] = \nabla(\Gamma_c \nabla k) + P - \rho\varepsilon \quad (6)$$

$$\rho[\vec{V} \cdot (\nabla.\varepsilon)] = \nabla(\Gamma_\varepsilon \nabla \varepsilon) + C_{\varepsilon 1} \frac{\varepsilon P}{K} - C_{\varepsilon 2} \frac{\rho \varepsilon^2}{k} \quad (7)$$

where the diffusion coefficients are given by

$$\Gamma_K = \mu + \frac{\mu_t}{\sigma_k} \quad (8)$$

and

$$\Gamma_\varepsilon = \mu + \frac{\mu_t}{\sigma_\varepsilon} \quad (9)$$

the turbulence viscosity, μ_t can be derived from Eq. (9) and (10), to link to the turbulence kinetic energy and dissipation via the relation

$$\mu_t = C_\mu \rho \frac{K^2}{\varepsilon} \quad (10)$$

and

$$\varepsilon = \frac{k^{\frac{3}{2}}}{l_t} \quad (11)$$

$$p_k = \mu_t \nabla U (\nabla U + \nabla U^T) - \frac{2}{3} \nabla U (\mu_t \nabla U + \rho k) \quad (12)$$

P_k is the term for the production of turbulence due to viscous forces Parameter values considered in the simulations are presented in the table 1.

Table 1: Parameters for modeling the k- ε model

C_μ	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	σ_K	σ_ε
0.09	1.45	1.9	1	1.03

Equations (1), (2), (6), and (7) form a closed set of nonlinear partial differential equations governing the fluid motion.

All the previous equations are valid both for the impeller and for the diffuser, however the rotational forces in the source terms will only apply for the impeller in the movement equation as a result of Coriolis forces and centrifugal forces

3.Pump, Geometry and Grid

With the three-dimensional model there is a useful approach for investigation of flow behavior in different parts of pump. Figure 2 shows the structured grid generated. There are 224160 cells in the impeller and 133611 cells unstructured in the volute. This is enough for precise boundary layer simulation and it gives correct values for the pump performance and allows to analysis the details of the main phenomena involved. Surface between inlet-impeller and impeller-volute correspond to grid interfaces. The multiple reference frame (MRF) technique allows the relative motion of the impeller grid with respect to the inlet and the volute during steady simulation. Grid faces do not need aligned on both sides. The original impeller has a specific speed of 32. The main pump (impeller + volute) parameters are presented in Table2. The pump impeller was modified adding splitter blades

Table.2: Geometrical parameters of the pump NS = 32.

Pump NS32		
Parameters	Value	Description
Impeller		
R_i	115 mm	Inlet flange radius
R_1	75 mm	Mean impeller inlet radius
b_1	85.9 mm	Inlet impeller width
β_1	70°	Inlet blade angle
θ_1	37°	Blade LE inclination angle
R_2	204.2 mm	Mean impeller outlet radius
b_2	42 mm	Outlet impeller width
β_2	63°	Outlet blade angle
θ_2	90°	Blade TE inclination angle
Na	5	Blade number
e	8 mm	Blade thickness
Volute		
R_3	218 mm	Base volute radius
b_3	50 mm	Volute width
ϕ_{outlet}	200 mm	Outlet flange diameter

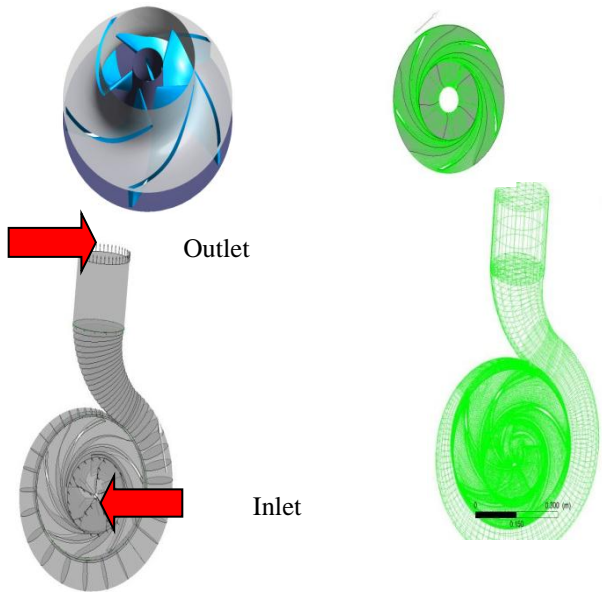


Figure 2 :shows Impeller structured , Volute unstructured the grid generated

4.1 Simulation parameters and boundary conditions

At the inlet of the computational domain an imposed pressure is specified (Table 2 and Fig. 2). At the outlet a mass flow q is imposed following the operating condition (Table4). A no-slip flow condition is applied on the walls (on the blade, hub and shroud). Due to the change between the reference frame of the static volute and rotating impeller, the interaction of impeller-volute has been simulated using the Frozen-Rotor interface model (Table3).

The original configuration of the outlet volute casing was modified in order to avoid the occurrence of a big recirculation in this area. The extension of the volute outlet is essential for numerical and physical reasons, since convergence problems and related flow instabilities are prevented if the swirling zone is captured into the simulation domain

Table 3 : Boundaries conditions of impeller in the pump NS32

Flow simulation domain	Impeller
Grid	Impeller structured,
Fluid	Water at standard conditions
Inlet	Total pressure = 1 atm
Outlet	Mass flow = q (kg/s)
Turbulence model	$k-\epsilon$ and SST
Convection scheme	second order
RMS (root mean square)	4-Oct

Table 4 : Value of different mass flow q/q_v

Value	0.5	0.8	1	1.2
-------	-----	-----	---	-----

4.2 Numerical Investigation

The transport equations associated with the given boundaries conditions describing the internal flow in centrifugal pump are solved by the CFX code. This code is based on the finite-volume method to discretize the transport equations (ANSYS-CFX, 2011). For all presented numerical results, the momentum and continuity equations are solved simultaneously. In CFX, the pressure and velocity coupling is solved using the Rhie-Chow algorithm. Second order high resolution scheme have been adopted for convection terms.

5. Results

5.1 Meridian velocity fields for different mass

Qualitatively, the meridional velocity profile shows a very satisfactory pace,. A meridional velocity uniform output impeller promotes the functioning of the volute or the rectifier located downstream of the impeller. All flow parameters are strongly influenced by the flow. In the study of a pump, an analysis of its behavior nominal outside is very important, especially in partial flow .Figure 3 shows the distribution of meridional velocity at 20% of rated flow.Note the appearance of a small perturbation to the leading edge at the front flange, reflecting a less stable velocity distribution for the machine. Although this information can be useful, it is necessary to use it with caution, since other part flow more complex phenomena arise in viscous fluid (separation, recirculation). Perfect fluid flows are not able to make realistic forecasts in this area. These phenomena are taken into account rigorously using CFD codes in the following pages. Nevertheless, the main structures of the flow as the high accelerations at the origin of the recirculation are highlighted in all flow of operation of the pump.

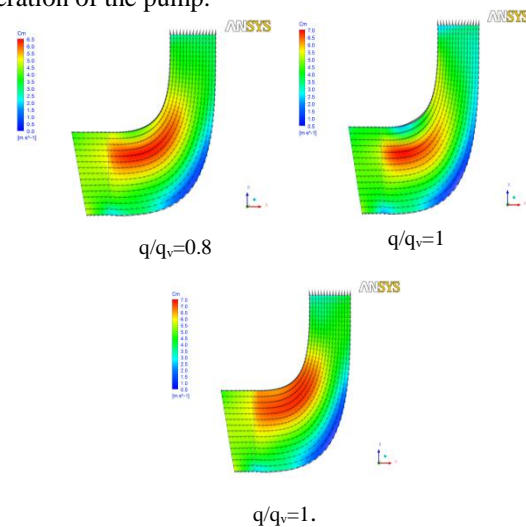


Figure 3; Meridian velocity fields for different mass flow

5.2 Blade loading at pressure and suction side

The blade loading of pressure and suction side are drawn at three different locations on the blade at the span of 20, 50 and 80 from hub towards the shroud. The pressure loading on the impeller blade is shown in figure 5 Pressure load on the impeller blade is plot along the streamwise direction. The pressure difference on the pressure and suction sides of the blade suggests that the flow inside the impeller experiences the shearing effects due to the pressure difference on blade-to-blade passage wall.

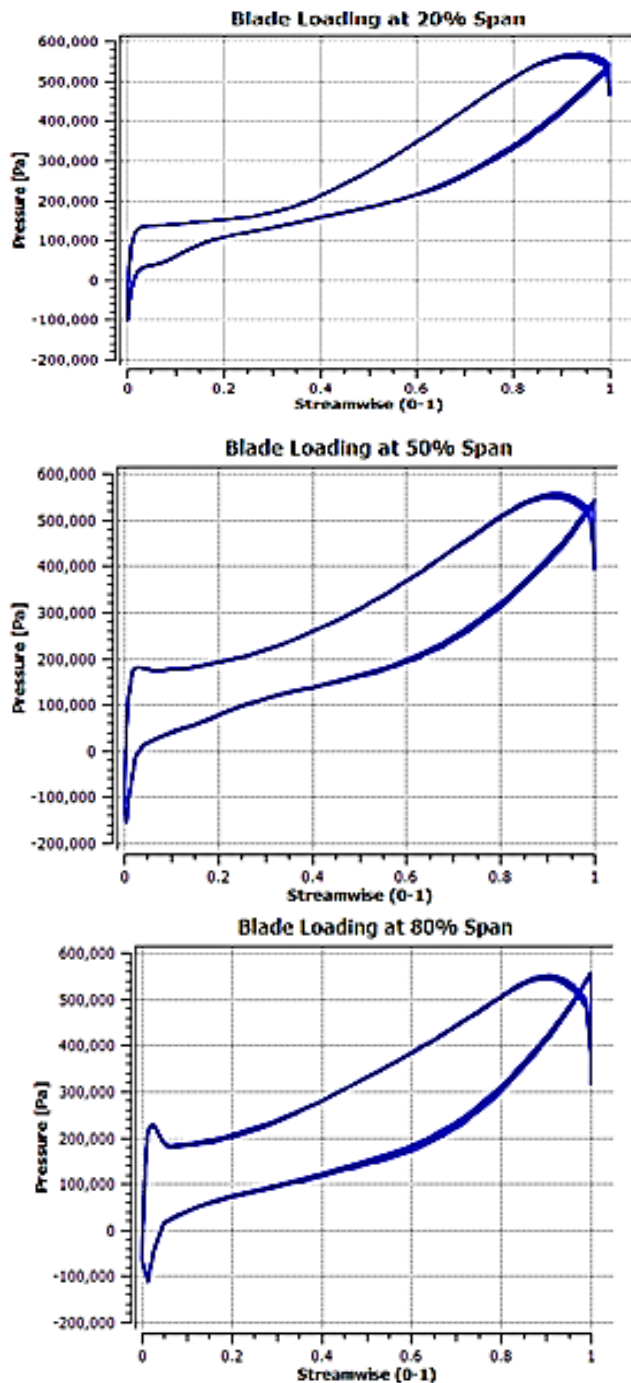


Figure4: Blade loading at 20, 50 and 80 span nominal mass flow ($q/q_v=1$)

5.3 field pressure for deferent mass flow in the impeller

The distribution of total pressure of the impeller for deferent mass flow is shown in the figure5. The lowest total pressure appears at the inlet of the impeller suction side. This is the position where cavitation often appears in the centrifugal pump. The highest total pressure occurs at the outlet of impeller, where the kinetic energy of flow reaches maximum. It is observed that the static pressure inside the impeller blades is asymmetry distributed. The minimum static pressure area appears at the back of the impeller blade ie suction side, at the inlet.

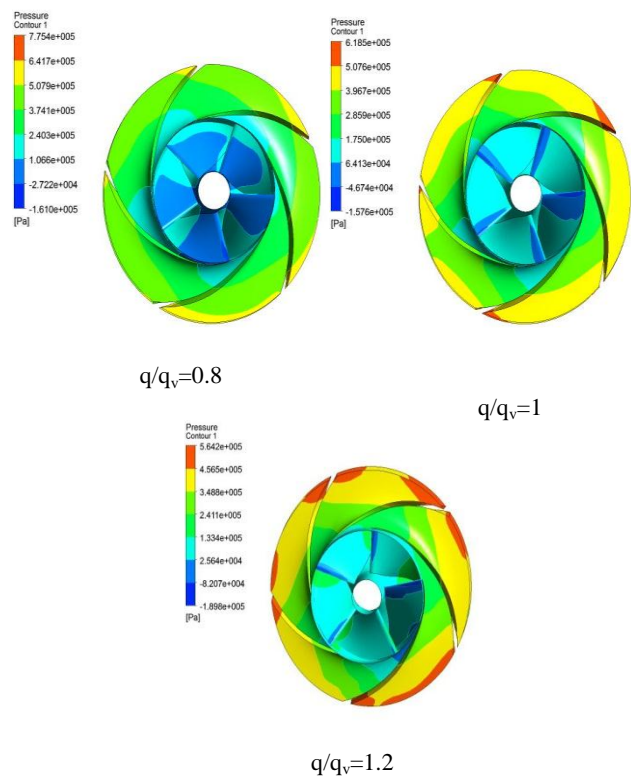


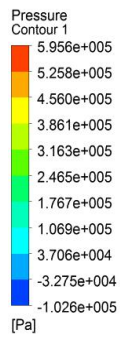
Figure5; pressure fields for different mass flow

5.4 field pressure pump for deferent mass flow

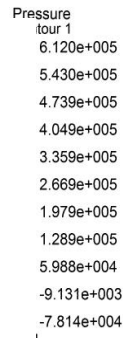
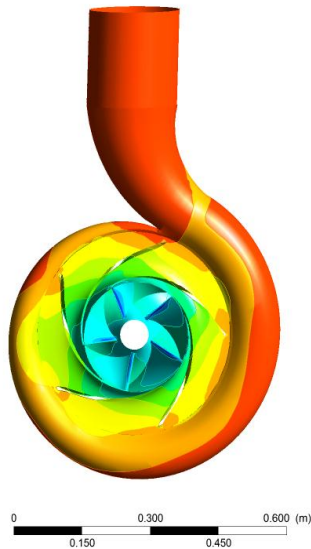
The figures6 shows the influence of flow on the kinematics of the general flow. We present, for the reference position ($\alpha = 0^\circ$), the static pressure field for four rates (0.5 Q_n , 0.8 Q_n , 1.0 Q_n 1.2 Q_n). At nominal flow, the fluid is properly channeled by the volute and we see a smooth operation of the wheel, the pressure field is slightly different in the various channels. By cons, greater variations are observed for the nominal operating points out.

5.5 field Velocity pump for deferent mass flow

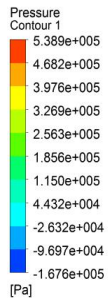
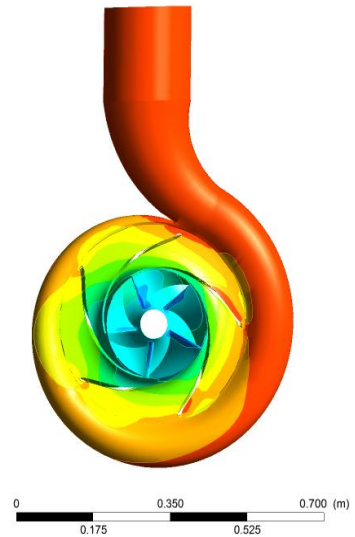
Figure 7 shows the influence of flow on the kinematics of the general flow. We present, for the reference position ($\alpha = 0^\circ$), the velocity vectors for four static discharge (0.5 Q_n , 0.8 Q_n , 1.0 Q_n 1.2 Q_n)



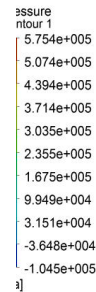
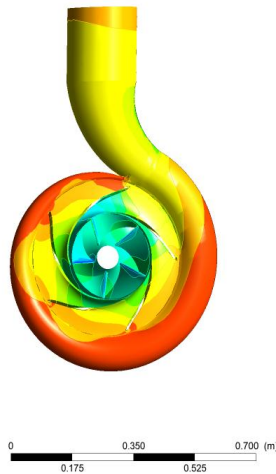
$q/q_v=0.8$



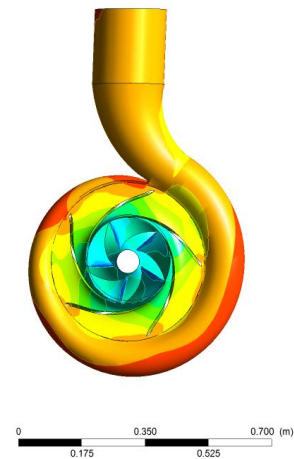
$q/q_v=0.5$



$q/q_v=1.2$



$q/q_v=1.$



Figures 6: Pressure fields for different mass flow

At nominal flow, the fluid is properly channeled by the volute and we see a smooth operation of the wheel, the velocity field is slightly different in the various channels. By cons, greater variations are observed for the nominal operating points out.

6. Conclusion

The commercially available 3D Navier-Stokes code called CFX which has a standard k-ε two-equation turbulence model, was chosen to simulate the internal flow of a commercial centrifugal pump. The complete centrifugal pump including impeller side gaps and discharge nozzle has been modeled and meshed with an unstructured grid. The predicted results are presented in terms of pressure profiles and velocity vectors. Having a closer look at the flow rates in a rotating impeller, the multiframe reference frame approach fails completely due to its fixed coupling formulation. However, the present approach is useful for basically understanding the flow states in various operating points. Certainly the scheduled transient calculations may serve as a real tool for understanding the interaction between impeller and spiral casing. Further work will focus on a detailed statement on the influence

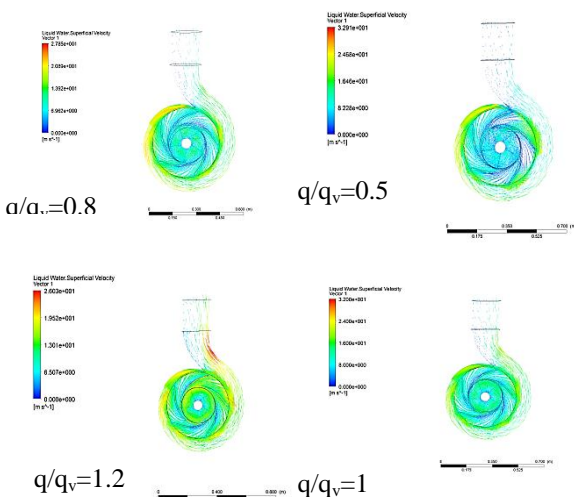


Figure 7: Velocity fields for different mass flow

References

Anagnostopoulos, J. S., (2006), Numerical calculation of the flow in a centrifugal pump impeller using cartesian grid

International Conference on Applied and Theoretical Mechanics, Venice, Italy, pp 134-154.

Baun, D. O. and Flack, R. D., (2003), Effects of volute design and number of impeller blades on lateral impeller forces and hydraulic performance, *International Journal of Rotating Machinery*, volume 9, pp 145–152

Cheah, K.W., Lee, T. S., Winoto, S. H., and Zhao, Z.M., (2007), Numerical flow simulation in a centrifugal pump at design and off-design conditions, *International Journal of Rotating Machinery*, Volume 27, pp 238-254.

Hergt, P., Meschkat, S. and Stoffel, B., (2004), The flow and head distribution within the volute of a centrifugal pump in comparison with the characteristics of the impeller without casing, *Journal of Computational and Applied Mechanics*, Volume 5, pp 275-285

Miner, S. M., (2001), 3-D Viscous flow analysis of a mixed flow pump impeller, *International Journal of Rotating Machinery*, Volume 7, pp 53—63

Majidi, K. and Siekmann H. E., (2000), Numerical calculation of secondary flow in pump volute and circular

casings using 3D viscous flow techniques, *International Journal of Rotating Machinery*, Volume 6, No. 4, pp 245-252.

Ogut, A. and Pastor, D. G., (2000), Simulation of flow in turbopump vaneless and vaned diffusers with fluid section, *International Journal of Rotating Machinery*, Volume 6, No. 1, pp 57-65.

Spence, R., Amaral-Teixeira, J., (2007), Investigation into pressure pulsations in a centrifugal pump using numerical methods supported by industrial tests *Computers & Fluids*, Article in Pres.

Younsi, M., Kergourlay, G., Bakir, F., and Rey, R., (2007), Influence of splitter blades on the flow field of a centrifugal pump: test-analysis comparison, *International Journal of Rotating Machinery*, Volume 07, Article ID 85024, 13 pages.

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