ISSN 1392–1207. MECHANIKA. 2015 Volume 21(4): 301–306

Numerical approach based design of centrifugal pump volute

A. Allali*, S. Belbachir, A. Lousdad***, L. Merahi******
**Laboratory of applied mechanics, Faculty of Mechanical Engineering, University of sciences and technology of Oran (USTO), B.P. 1505, Oran El M'Naouer, 31000, Oran, Algeria, E-mail: ahmedenergetique @gmail.com*
***LMPM, Mechanical Engineering Department, Faculty of technology, University of Sidi Bel Abbes, BP 89 Cité Ben M'hidi, Sidi Bel Abbes, 22000, Algeria, E-mail: belbachirsaida@yahoo.com*
****Laboratoty mechanics of structure and solids (LMSS), Mechanical engineering department, Faculty of technology, University of Sidi Bel Abbes, 22000 Sidi Bel Abbes, Algeria, Email: a-lousdad@yahoo.com*
*****Laboratoty of applied mechanics, faculty of Mechanical Engineering, University of sciences and technology of Oran (USTO), B.P. 1505, Oran El M'Naouer, 31000, Oran, Algeria, E-mail: l_merahi@yahoo.fr*

crossref http://dx.doi.org/10.5755/j01.mech.21.3.8930

1. Introduction

Centrifugal pumps are widely used in different domains of industry. They belong to the family of turbo-machinery. The main role of pumps is to transform mechanical energy generated by motors to hydraulic energy. Each particle of the fluid away form the axis of rotation its trajectory is referred to as centrifugal. It is necessary, for a better understanding, to know the way of design of three pumps in order to improve and increase their energetic efficiency and performance by minimizing the losses. In this point of view, the manufacture must have precise, rapid and reliable methods for the contribution of a wheel or volute. In this regards, the main objective of the present work is to make a contribution to the research carried out in this domain by applying the laws governing the nature of internal flow. This concerns both, the Newtonian and Non Newtonian fluid flow in the volute of the pump. A numerical analysis has been carried out to investigate the properties of the internal flow in three different forms of type of volutes namely circular, Bezier and trapezoidal with the same dimensions and same rotor. The commercial computer code ANSY-CFX is used to perform the numerical analysis by implementing the finite volume method. From literature, it has been noticed that most of the research focus on the prediction of centrifugal pump performance with radial volute and rotor dimensioning. K. FIsher and D. Thoma [1] are among the first researchers that had observed a difference between the fluid flow properties for real fluid and perfect fluid in bladed wheel. The field of unstable three dimensional fluid flow of multi-stage turbine had been numerically investigated by Rai [2] (1987and 1989). Otherwise, Bakir et al. [3] were oriented toward experimental studies for improving resistance to cavitations of axial inductors. A re-dimensioning and optimisation of an oxygen liquid pump of Vulcan motor were studied and worked on by Arnone and al. [4]. Recently, Aghaali et al [5] were interested in performance characteristics of radial turbine of a double input turbocompressor with an asymmetric volute at stationary state taking into consideration the complete and partial admission conditions. Sloteman et al. [6] had developed a design methodology based on the combination of one-dimensional analysis method of performance and an inverse method for blade generation. An automatic procedure relying on the combination of different analysis and approaches for dimensioning was proposed by Cravero [7]. The dimensioning of hydraulic components of the pumps was numerised by Goto et al., [8]. Chiong et al [9] had introduced in their numerical one-dimensional studies the conditions of pulsed flow and effect of geometry in double input compressor turbine with complete admission. Currently, the use of dimensioning and analysis codes for no-viscous flow a 3D and a quasi 3D geometrical modifications were proposed by Paβruker and Van den Braembussche [10].

2. Theory of internal flow in centrifugal pumps

2.1. Equations of incompressible flow

The characteristics of an incompressible flow [11] are obtained by using the following equations:

The continuity equation in the fixed reference system is given by:

$$\vec{\nabla}\vec{C} = 0. \tag{1}$$

Navier-Stokes equation in the fixed reference system which can be written as:

$$\rho\frac{D\vec{C}}{Dt} = -\vec{\nabla}p + \rho\vec{F} + \mu\vec{\nabla}^2\vec{C}, \tag{2}$$

where p is the pressure, $\vec{F}$ is the force, μ is the dynamic viscosity, ρ is the density and t is the time.

2.2. Equations in relative reference system

The vectorial composition of the velocities inside the centrifugal pump is given by:

$$\vec{C} = \vec{W} + \vec{U}, \tag{3}$$

where $\vec{U}$ is the drawing velocity, $\vec{C}$ is absolute velocity and $\vec{W}$ is relative velocity.

$$\vec{U} = \vec{\Omega}\vec{r}, \tag{4}$$

where Ω is angular velocity of the pump.

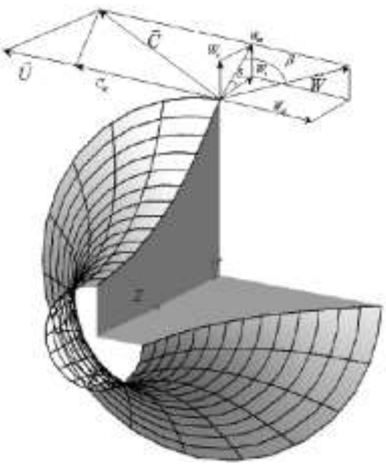

Fig. 1 Flow in three dimensional grid

The relation between the components of the different velocities is shown in (Fig. 2):

$$\begin{cases} C_r = W_r; \\ C_u = W_u + \vec{\omega}\,\vec{r}; \\ C_z = W_z. \end{cases} \tag{5}$$

x, y, z Cartesian coordinates; r, θ, z Cylindrical coordinates

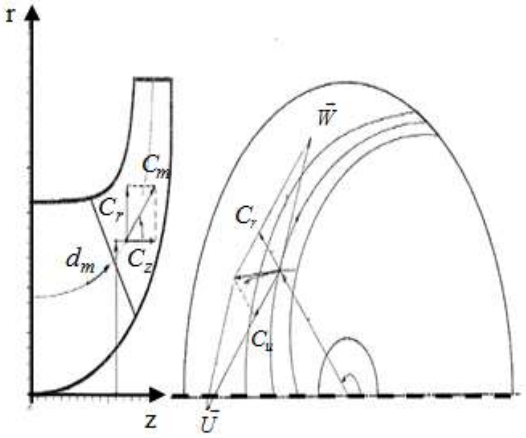

Fig. 2 Velocity triangles. Meridian view and Face view

The continuity equation of relative movement is given by:

$$\frac{\partial \rho}{\partial t} + \nabla\left(\rho \vec{W}\right) = 0\,. \tag{6}$$

The equation of fluid displacement in relative reference system is:

$$\frac{D\vec{W}}{Dt} + 2\vec{\Omega}\vec{W} + \vec{\Omega}\,\vec{\Omega}\,\vec{r} = -\frac{\nabla P}{\rho} + \vec{F} + \\ + \frac{1}{\rho}\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial W_i}{\partial x_j} + \frac{\partial W_j}{\partial x_i}\right) + \delta_{ij}\lambda div\vec{W}\right], \tag{7}$$

where λ is thermal conductivity.

2.3. Plot of volute according to the principal of kinetic moment

Based on the assumption of irrotational fluid flow in the volute [12] we have:

$$\chi = Cu\,r = Cu_2\,R_2\,, \tag{8}$$

with R_2 is rotor radius and χ is kinetic moment.

The section element is given by:

$$dA = b(r)\,dr\,. \tag{9}$$

The flow rate is equal to:

$$dq_v = Cu\,dA\,, \tag{10}$$

with *dA* is the cross section element, q_v is flow rate and *b* is volute width.

By integrating along the whole section we have:

$$q_{v\theta} = \chi \int_{R_3}^{R_\theta} b(r)\frac{dr}{r}\,, \tag{11}$$

where R_3 is radius of the volute base and R_θ is spiral radius.

The fraction of total flow rate in pump is equal to:

$$q_{v\theta} = \frac{\theta}{360} q_v\,. \tag{12}$$

2.4. No-Newtonian fluid model

There exists adjusting mathematical models of reo-routing available in the majority of reo-treatment codes. These models are expressed under the form of shear stress with respect to shear ratio [13]:

$$\tau = f(\dot{\gamma}). \tag{13}$$

The viscosity in all cases is defined as:

$$\mu(\dot{\gamma}) = \frac{f(\dot{\gamma})}{\dot{\gamma}}\,. \tag{14}$$

The model used in this study is the power law model:

$$\begin{cases} \tau = K\dot{\gamma}^n; \\ \mu = K\dot{\gamma}^{n-1}, \end{cases} \tag{15}$$

with τ is tangential stress, $\dot{\gamma}$ is shear rate and K is constant.

3. Types of volute configuration studied

In this work a Newtonian and No-Newtonian internal liquid flow with three different volute configuration have been investigated. (Fig. 3) show the volute configuration under study. The aim is to know the effect of volute geometry and the nature of fluid on the flow parameters (velocity, total input pressure and shear stress). The conduct allows having a developed flow at the centrifugal pump inlet. The wall of configuration shown is assumed to be undeformable and do not favour sliding. The internal liquid flow is assumed to be three-dimensional, turbulent and steady with inlet pressure of

101325 Pa and an outlet flow of 163.8 kg/s. The computation of these parameters is based on Navier-Stokes and conservation equations. The calculation is carried out using ANSYS code. In the case where the fluid is considered Newtonian water has been used. If the fluid is No-Newtonian the power law model is taken into account by means of subprogram. The study of the flow parameters is carried out for three different tangential volute types (circular, Bezier and trapezoidal) with the same dimensions and same rotor. In each volute the evolution of the velocity magnitude, the pressure and the shear stresses are computed for Newtonian and No-Newtonian cases.

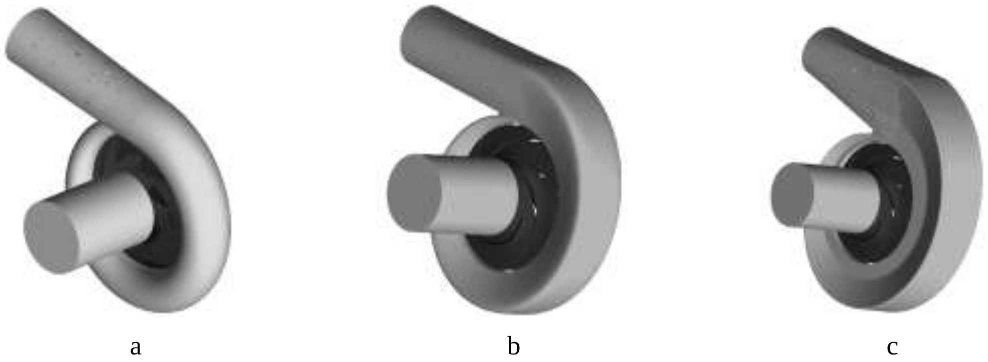

Fig. 3 Volute Configuration: a - circular volute; b - bezier volute; c - trapezoidal volute

4. Meshing of the pump

The meshing of the pump must be conform with the physical and geometrical criteria and takes into consideration constraints relative to the study (the sense of the flow, the boundary conditions, definition of the interface …etc.).The mesh is made using ANSYS ICEM – CFD It is of tetrahedral type structured for the wheel and unstructured for the conduct and volute (Fig. 4).

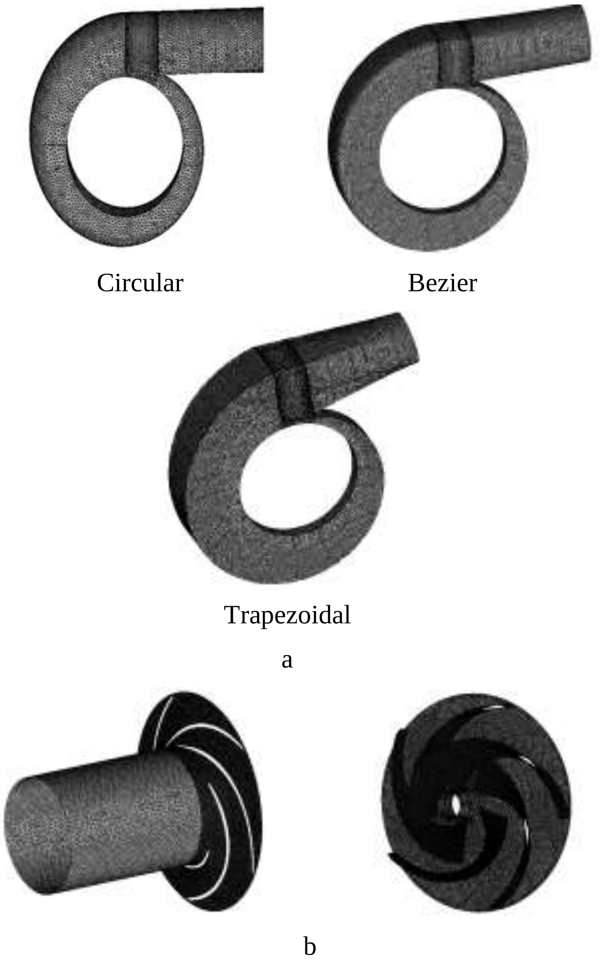

Fig. 4 Mesh of centrifugal pump: a - volute; b - conduct and hub; c - wheel

5. Conditions of simulation

With the CFX-Pre module, the conditions of simulation have been introduced whose characteristics are grouped in Table:

Table

Characteristics of meshing

Characteristics	CFX-5
Simulation domain	impeller (5 blades) + volute
Mesh of conduct	Non-structured
Mesh of impeller	Structured
Mesh of volute	Non-structured
Input condition	Total Pressure = 101325 Pa
Conduit/impeller interface	Frozen-rotor
Impeller/volute interface	Frozen-rotor
Output condition	Flow rate: 163.8 Kg/s
Turbulence model	k-Ɛ
Numerical scheme	Second order
Average residu (RSM)	10^{-4}

6. Results and discussion

The obtained results of total pressure and velocity magnitudes were grouped for comparison. In each volute, the evolution of velocity magnitude, total pressure and shear stress are calculated for Newtonian and No-Newtonian case as illustrated on (Figs. 5-12).

6.1. Evolution of total pressure

The internal flow of Newtonian fluid in the centrifugal pump is more complex due to the recirculation generated in Bezier and circular volutes and their conjugated effects. This is shown by the negative values of the total pressure (Figs. 5 and 6). The depression shows the presence of a recirculation which is due to the taking off of the boundary layer. It is observed that there is always a depression zone at the extrados. The pressure distribution is not identical in all passages of the volute. The pressure is non homogenous as observed at the bec zone. It is characterised by a high pressure gradient. The form of the volutes creates a geometric dissymmetry which affects the pressure field in the pump.

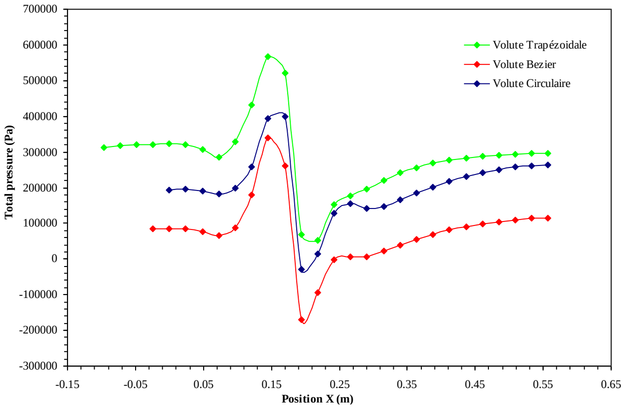

Fig. 5 Evolution of total pressure at position (X) Fluid Newtonian

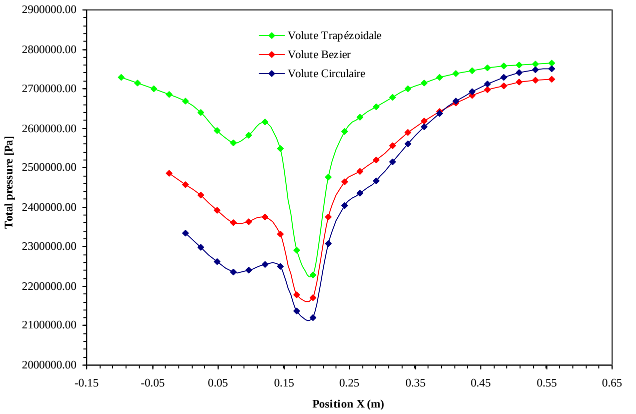

Fig. 6 Evolution of total pressure at position (X) Fluid no-Newtonian

Concerning the effect of the nature of the fluid (Newtonian or no- Newtonian), there exists a significant gap observed in the total pressure evolution. There is no recirculation in the case of no-Newtonian fluid (Figs. 6, 7 and 8). It is observed that the effect of recirculation is more apparent in the Bezier volute and less sensitive in circular volute. This effect does not exist in trapezoidal volute. The pressure has a characteristic behaviour of a centrifugal pump because the volute transforms the kinetic energy under the form of pressure energy. The higher values are found in the volute at the outlet and the lower values are at the inlet geometry.

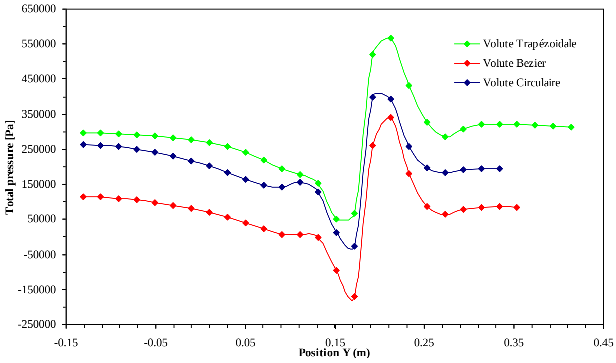

Fig. 7 Evolution of total pressure at position (Y) Fluid Newtonian

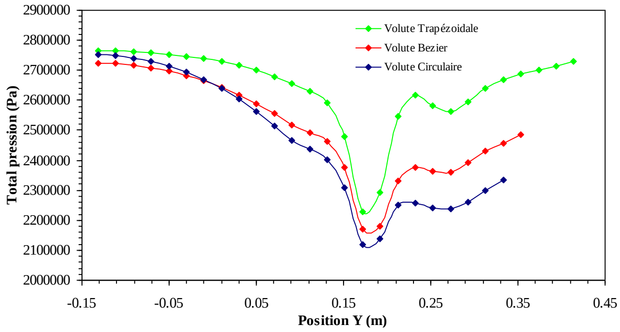

Fig. 8 Evolution of total pressure at position (Y) Fluid no-Newtonian

6.2. Evolution of the velocity magnitude

The magnitudes of the velocity decrease due to the wall geometry of the volute presenting a curve whose center is on fluid side, and incurved in the same sense as the stream lines.

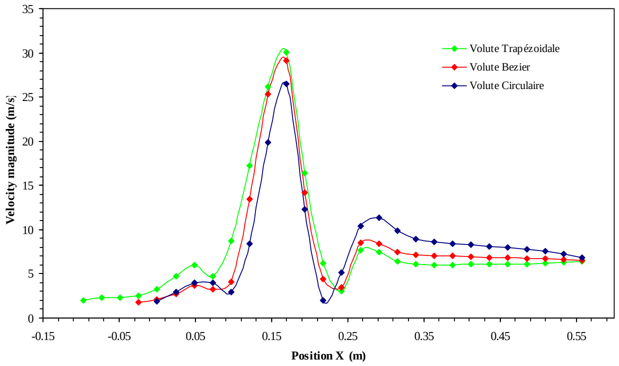

Fig. 9 Evolution of velocity magnitude at position (X) Fluid Newtonian

The boundary layer, which represents the thin layer of the fluid, is influenced by the contact with the wall. Once the section is uniform in the volute, it is noticed an important velocity gradient between the wall and the point of maximum velocity. This is confirmed by the evolution and increase of the pressure.

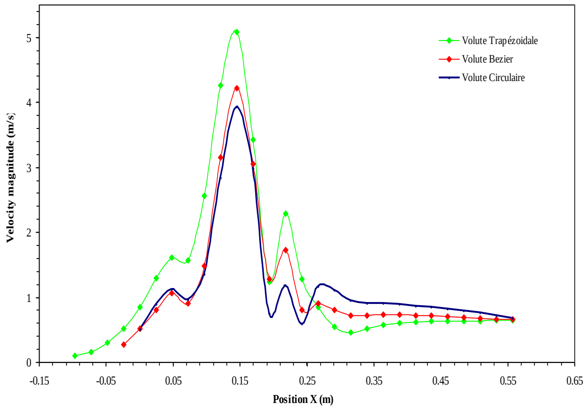

Fig. 10 Evolution of velocity magnitude at position (X) Fluid no- Newtonian

On the other hand, due to the geometry of each investigated volute, the magnitudes of the velocities are not equal. However, the form of the plots is similar in the three cases of the studied volutes (Figs. 9-12).

It is observed an increase of the velocity magnitude in the case of a Newtonian fluid and decrease in the case of No-Newtonian fluid. The recirculation phenomenon of liquid particles generates a considerable loss in velocity at the volute outlet.

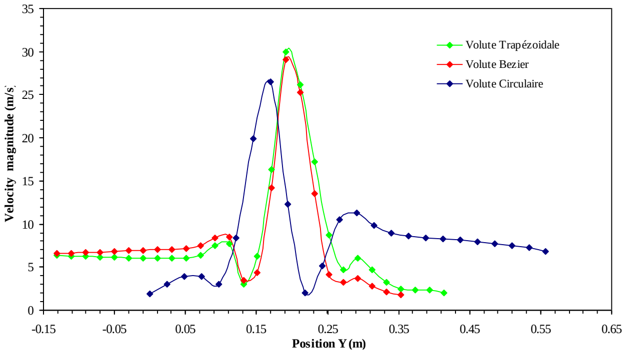

Fig. 11 Evolution of velocity magnitude at position (Y) Fluid Newtonian

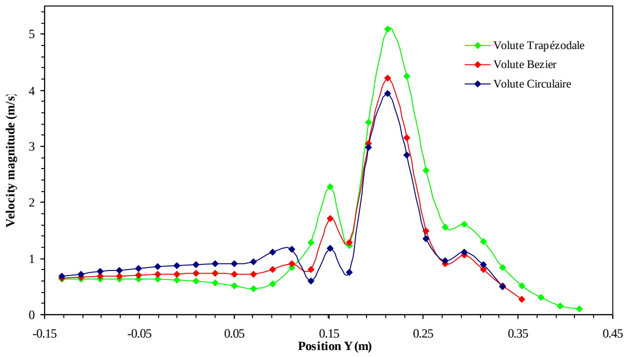

Fig. 12 Evolution of velocity magnitude at position (Y) Fluid no- Newtonian

6.3. Shear stress distribution

The Shear stress contours are illustrated in (Fig. 13) left and right side for Newtonian fluid and No-Newtonian fluid in the three volutes in order to show the stress distribution in the centrifugal pump made of the wheel and volute. It is observed that the values of the shear stresses in the three volutes for the case of a Newtonian fluid are lower than those of No- Newtonian fluid.

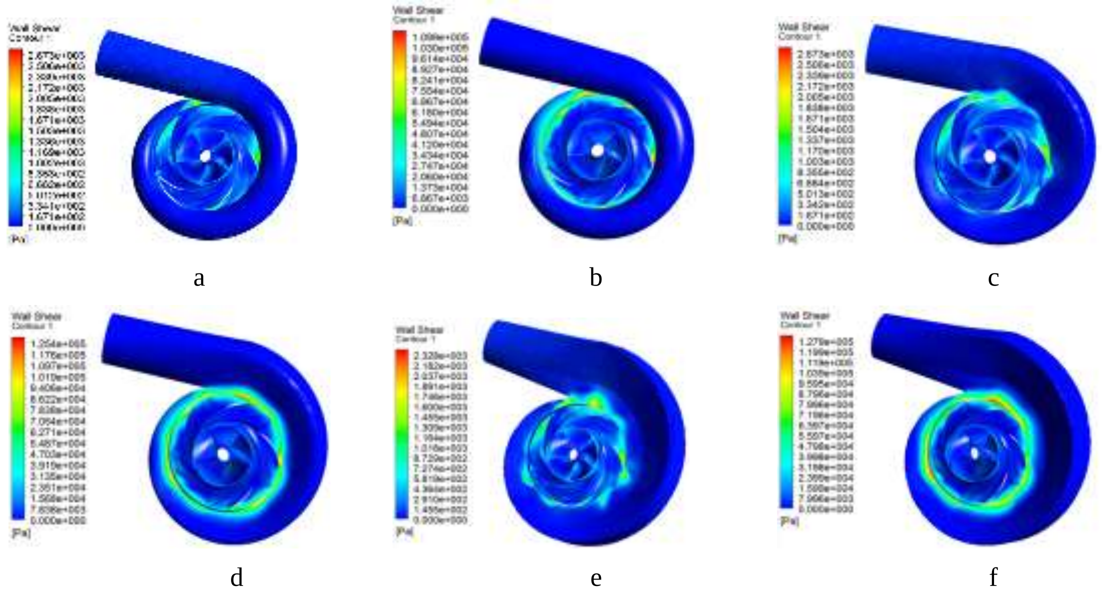

Fig. 13 Contour display of shear stresses: a - Newtonian Fluid in Circular Volute; b - No-Newtonian Fluid in Circular Volute; c - Newtonian Fluid in Bezier Volute; d - No-Newtonian Fluid in Bezier Volute; e - Newtonian Fluid in Trapezoidal Volute; f - No-Newtonian Fluid in Trapezoidal Volute

In General, the shear stresses are not uniformly distributed and are higher in the wheel mainly at the inlet of the volute. The value of the shear stress at the inlet of the volute is higher than at the remaining part of the volute, since the velocity gradient which is relatively higher in the inlet side of the volute. For the case of No-Newtonian fluid, the evolution is similar but with lower values due to the decrease of the viscosity when the shear ratio increases (power law model).

7. Conclusion

The objective of the present work concerns the numerical study of internal fluid flow in a centrifugal pump and the behaviour of three different forms of volutes under pressure load. The case of Newtonian fluid (water) and no-Newtonian (using power law model) are examined in order to see their effects on the properties and parameters of internal flow. The computation were carried out by ANSYS code. The results show that the geometry of each volute as well as the nature of the fluid have a significant effect on the velocity of the flow, on total pressure and shear stresses. A gap is observed between the shear stresses for two cases and was put into evidence. It is relatively higher in the zones with higher velocity gradient due to the decrease viscosity of the no-Newtonian fluid. This later is more resistant to the formation of recirculation zones generated by the presence of elbows in the volutes under study. This shows that when the viscosity is higher in the fluid the recirculation zones are lower. It may be concluded that the parameters (pressure and velocity) of the fluid flow are sensitive to the geometry variations and the fluid contrarily to the shear stresses. The obtained numerical results of the design of turbo machinery are in good agreement with the experimental observations and results found in the literature.

References

1. **Fisher, K.; Thoma, D.** 1932. Investigation of the flow conditions in a centrifugal pump, Trans. ASME HYD-54-8: 141-155.
2. **Rai, M.M., Madavan, N.K.** 1990. Multi-airfoil Navier-Stokes simulations of turbine rotor-stator interaction, J. Turbom. 112: 377-384. http://dx.doi.org/10.1115/1.2927670
3. **Bakir, F.; Kouidri, S.; Noguera, R.; Rey, R.** 2003, Experimental analysis of an axial inducer: influence of the shape of the blade leading edge on the performances in cavitating regime, ASME Journal of Fluids Engineering 125: 293-301. http://dx.doi.org/10.1115/1.1539872.
4. **Arnone, A.; Bonicelli, P.; Munari, A.; Spano, E.** 1999. Application of CFD Techniques to the design of the Ariane 5 Turbopump, 14th AIAA Computational Fluid Dynamics Conference, 1087-1097. http://dx.doi.org/10.2514/6.1999-3380.
5. **Aghaali, H.; Benisi, A.H.** 2007. Experimental and theoretical investigation of twin-entry radial inflow gas turbine with unsymmetrical volute under full and partial admission conditions, Proceeding of GT 2007 ASME Turbo Expo 2007, Power for Land ,Sea and Air http://dx.doi.org/10.1115/GT2007-27807.
6. **Sloteman, D.; Saad, A.; Cooper, P.** 2001. Designing custom pump hydraulics using traditional methods, Proceedings of ASME FEDSM01, paper FEDSM2001 -18067, New Orleans, Louisiana, USA, 2: 205-214.
7. **Cravero, C.** 2001. A design methodology for radial turbomachinery. Application to turbines and compressors, Proceedings of ASME FEDSM02, paper FEDSM2002 - 31335, Montreal, Quebec, Canada. http://dx.doi.org/10.1115/FEDSM2002-31335.
8. **Goto, A.; Zahgeneh, M.** 2002. Hydrodynamic design of pump diffuser using inverse design method and CFD, Transactions of the ASME, Journal of Fluids Engineering 124: 319-328. http://dx.doi.org/10.1115/1.1467599.
9. **Chiong, M.S.; Rajoo, S.; Botas, M.; Costallca, A.W.** 2012. Engine turbocharger performance prediction: one-dimensional modelling of a twin entry turbine energy conversion and management 57: 68-78. http://dx.doi.org/10.1016/j.enconman.2011.12.001.
10. **Paβruker, H.; Van den Braembussche, R.A.** 2000, Inverse design of centrifugal impellers by simultaneous modification of blade shape and meridional contour, in Proceedings of the 45th ASME International Gas Turbine & Aeroengine Congress, Munich, Germany. http://dx.doi.org/10.1115/2000-GT-0457.
11. **Asuaje, M.** 2003. Methodology and optimisation in the design and analysis of turbomachinery performance with incompressible fluid, Doctorate thesis: ENSAM-Paris.
12. **Rey, R.; Bakir, F.** 2008. Helicocentrifugal and centrifugal Machines ,T. III.
13. **Zaynab Salloum** 2010. Mathematical study of viscoelastic fluid flow in singular domains, Doctorate thesis - Paris-East University.

A. Allali, S. Belbachir, A. Lousdad, L. Merahi

NUMERICAL APPROACH BASED DESIGN OF CENTRIFUGAL PUMP VOLUTE

Summary

Centrifugal pumps are widely used in many industrial sectors. However the experimental characterisation processus of pumps is very complex and expensive task due to the large number of geometrical parameters in consideration. The complexity is increased due to their elaboration by molding process dicted by the pump components (volute, wheel, and rotor). It is then worthful to minimizing the costs associated to design and manufacturing processes of pumps. Ever though the tremendous efforts made in this domain the experimentation does not represent the most reliable solution. A precise numerical simulation of complex liquid flow in centrifugal pump can be rapid and reliable solution for improving the pump performance. A large number of numerical or experimental researches are focussed on rotor dimensioning of radial volute. In this study three different forms of tangential volutes (circular, Bezier and trapezoidal) with same dimensions and the same rotor are used. The behaviour of the fluid either Newtonian or Non-Newtonian is taken into consideration. The objective of the study is to know the effect of the form of the volute and nature of the fluid on the properties of the internal fluid flow. Finite volume method (FVM) is used under the commercial code Ansys-CFX. The numerical simulation is carried out to study the velocity distribution and pressure field distribution in the three volutes. The nature of Non-Newtonian fluid behaviour is integrated in the code via a subprogram. The turbulence of flows in the centrifugal pump is taken into account using k-Ɛ model. The obtain results show that the form of the volute and the nature of fluid flow have a sensitive effects on the velocity, the pressure as well as on the shear stress.

Keywords: centrifugal pump, volute, Newtonian and no-Newtonian fluid, k-ε turbulence model, shear stress, Ansys-CFX.

Received December 17, 2014
Accepted May 06, 2015