Impeller Geometry Effects on Performance of Centrifugal Pump for Handling Viscous Fluids

Mohammad Hassan ShojaeeFard1, Mojtaba Tahani2, Hadi Fallon3, Mehdi Bighlari4, Biyok Ehghaghi5

1 Department of Mechanical Engineering, IUST, Tehran, Iran; mhshf@iust.ac.ir
2 Department of Mechanical Engineering, IUST, Tehran, Iran; tahani@iust.ac.ir
3 Department of Mechanical Engineering, IUST, Tehran, Iran; h_fallah@mecheng.iust.ac.ir
4 Department of Mechanical Engineering, IUST, Tehran, Iran; mehdi_beglari@mecheng.iust.ac.ir
5 Department of Mechanical Engineering, Tabriz University, Iran; b_ehghaghi@yahoo.com

Abstract
Considering that centrifugal pump performance severely decreases in handling viscous fluids, so the reduction of performance is improved by variation in original geometry of centrifugal pump impeller. The present work is an attempt to simulate three-dimensional flow numerically from the centrifugal pump along with its volute, including original geometry and applied changes in geometry. The simulated centrifugal pump type is a 65-200, single axial suction and vane less volute casing, equipped with an impeller of 203 mm in outer diameter and 6 backwards curved blades. When the pump is running at 1450 rev/min and handling oil with 43Cst viscosity, the best efficiency point corresponds to 56 m3/h flow rate and 13.5 m height. A one dimensional theory model was established to identify such effect mathematically. The results showed that the blade exit angle is significant. The impeller with the exit angle of 30 degree and the passage width of 21 mm illustrates the best efficiency. Moreover, a one dimensional hydraulic model was proposed to investigate the effect of blade exit angle and passage width analytically at various operation conditions. The numerical simulation was done for three different blade exit angles 27.5, 30 and 32.5 degree with two widths of impeller passage 17 and 21 mm. Finite volume method was used for discrediting governing equations and solving equation was derived by a high resolution algorithm. SST turbulence model was used here. Steady state simulation uses a Multi reference frame in which impeller is in rotational reference frame, and volute is in stationary reference frame and connects to each other by frozen rotor. Numerical results have been published with experimental results and there was good agreement between these results. It can be seen that at blade exit angle 30 degree and 21 mm of passage width, head and efficiency increase in modified geometry of pump that and this is due to decreasing the dissipations arising by vortex formation in impeller passage.

Keywords: centrifugal pump; viscous fluid; outlet angle; passage width; performance; simulation; turbulence; SST

Introduction
The performance of centrifugal pump is impaired, in comparison with handling water, and due to increasing losses when it services a fluid of high viscosity. Viscosity is such as a resistor to pouring high viscous fluids influence to centrifugal pump performance by increasing the power, reducing the flow rate, head and efficiency, and making trouble for mechanical seals and causing more loads on bearings.

Many researches, such as Daugherty [1], Stepanoff [2], Telow [3], Ippen [4], and Itaya and Nishikawa [5], have been tested the performance of centrifugal oil pumps as a function of viscous oil viscosity. They proposed some correction factors when pump handles the viscous fluid for determining the performance. These typical results provided great insights into the effects of oil viscosity on the performance of centrifugal oil pumps, which have been applied in the important guidelines used today for the design of these pumps. Stoffel [6], Li [7], Li and Hu [8] conducted experiments using commercial centrifugal oil pumps.

Theoretical analysis of viscous fluid flow through turbo-machines is very difficult, and it needs more time for solving the equations by use of super computers. Within the last few years, several computer codes have been published for analyzing the effects of the viscosity and boundary layers on turbo machinery performance. The codes use different numerical schemes, boundary conditions and turbulence models for predicting the viscosity effects, impeller tip leakage and secondary flow through the blade row. By using this code, Denton [9] simulated the viscous effects in the passage of blades by use of a distributed body force. Miner [10, 11] has done several researches on two dimensional flow analysis and turbulence measurements of a centrifugal pump.

Gugau [12] simulated the flow in the centrifugal pump when handling water, by using the sliding meshes for analyzing the main existent phenomena in these pumps. Weidong Z. [13], Investigated the flow through centrifugal pump with four direct impellers when handling water by using computational fluid dynamics. Kitano Majidi [14] investigated unsteady flow in a centrifugal pump and it was shown that due to the interaction between impeller blades and the tongue of the volute casing the flow is characterized by pressure fluctuations, which are strong at the impeller outlet and in the vicinity of the tongue. Kergourlay G [15] studied the influence of splitter blades on the flow field of a centrifugal pump. Cheah K.W. [16] simulated the water flow in a centrifugal pump at design and Off-Design conditions. Xianwu, Luo [17] investigated the impeller inlet geometry effect on the performance improvement for centrifugal pumps. Grapsas V. et al. [18] conducted a

In this study at first the experimental tests on the centrifugal pump performances are performed with different blade outlet angles and impeller passage widths. Then the three-dimensional fluid flows are simulated by commercial code, when the pump is handling viscous oil.

The purpose of the present study is to show a numerical study of a centrifugal pump taking into account the whole 3D geometry of pump performances with different blade outlet angles and impeller passage heights which is used in experiments and solving the 3D Navier-Stockes equations using the finite volume method for occurrence of numerical simulation. The numerical results are validated with the experimental characteristic curve.

**Pump specifications**
The simulated centrifugal pump is a 65-200, single axial suction and vane less volute casing, equipped with an impeller of 209 mm in outside diameter and 6 backwards curved blades. The blade outlet and wrapping angles of the impeller are 27.5° and 140°, respectively. When the pump is run at 1450 rev/min, the best efficiency point (BEP) corresponds to 56 m^3/h flow rate and 13 m height.

The impeller made of metal is machined. The roughness of the impeller and volute is 100 μm. The three-dimensional schematic of the given pump and its volute and outlet pipe are shown in Figure 1.

The meridional plane of main geometry of impeller and modified geometry in passage width are shown in Figure 2.

The centrifugal pump impeller is simulated for six different geometries. The outlet blade angles are 27.5, 30 and 32.5 degree, and the width of impeller passage are 17 and 21 mm, as shown in figure 2 and figure 3.

**Working fluid**
The centrifugal pump performance and flow fields in the impeller are measured using the special transparent viscous oil refined from crude oils as a working fluid.

They are Newtonian fluids verified by using the rotary viscosity meter. The density and kinematic viscosity of oil at 20°C, are 875 kg/m^3 and 42 mm^2/s, respectively.

**Numerical scheme**
The discipline of computational fluid dynamic (CFD) has fostered a unified approach to turbomachinery analysis and design. In turbo-machines the geometry is highly three-dimensional. Since the transfer of energy is brought about through the centrifugal forces that the impeller passes on to fluid, and this forces have radial and tangential directions, it is possible to consider that the model captures the fundamental phenomena that occur in pump.

For these numerical simulations the High-Resolution algorithm is used for solving governing equations of incompressible viscous/turbulent flows through the pump at different operating conditions. The SST k-ω turbulence model is adopted to describe the turbulent flow process. The feasibility of the proposed model will be discussed later.

Commercial code provided powerful set of features available for solving problems in which fluid rotates around an axis, such as flows inside turbomachines in different methods. The multiple reference frames (MRF) model is appropriate for steady flows and for cases in which the interactions between rotor and stator are negligible. For instance, for a pump with bladeless volute the MRF can be used as a suitable approach. Since in the present problem, the volute has no vanes, the simulations have been made with a steady calculation using the multiple reference frames (MRF) technique to take into account the impeller - volute interaction.

With the three-dimensional model there is a useful approach for investigation of flow behavior in different parts of pump and operating condition.

Figure 4 shows the unstructured grid generated. There are 1284666 cells in the impeller zone, 557816 cells in the volute and outlet pipe. This is enough for precise boundary layer simulation and it gives correct values for the experimental pump performance and allows to analysis details of the main phenomena involved.

Surface between impeller-volute and volute-pipe correspond to grid interfaces. The MRF technique
allows the relative motion of the impeller grid with respect to the impeller and the volute during steady simulation. Grid faces do not need aligned on both sides.

Continuity and 3-D incompressible Navier-Stockes equations, including the centrifugal force source in the impeller and steady terms are used in centrifugal pump to analyzing the turbulent viscous fluid flow. The High Resolution algorithm discretizations have been used for convection terms and use a form of the Gauss’ divergence theorem for diffusion terms.

Continuity and 3-D incompressible Navier-Stockes equations, including the centrifugal force source in the impeller and steady terms are used in centrifugal pump to analyzing the turbulent viscous fluid flow. The High Resolution algorithm discretizations have been used for convection terms and use a form of the Gauss’ divergence theorem for diffusion terms.

Continuity and momentum equation of an incompressible flow are as follows:

\[ \nabla \cdot \mathbf{u} = 0 \]  
\[ \nabla (\rho \mathbf{u}) = -\nabla p + \nabla \tau + s \]

In the above equations \( \mathbf{u} \) is the relative velocity of fluid, \( \tau \) is the stress tensor and \( s \) is the source term, which consists of Coriolis and centrifugal forces

\[ s = -2\rho \Omega \times \mathbf{u} - \rho \mathbf{u} \times (\Omega \times \mathbf{r}) \]

Here \( \Omega \) is the rotational speed and \( \mathbf{r} \) is the position vector.

**Boundary conditions**

At the inlet, a Stationary Frame Total Pressure is imposed. This is the same as the Total Pressure condition in a stationary domain. In a rotating domain, the total pressure is based on stationary frame conditions. The turbulence intensity value of 0.05 (that is, 5%) is sufficient for nominal turbulence through a circular inlet, and is a good estimate in the absence of experimental data. At the outlet, mass flow rate is maintained. Non slip boundary conditions have been imposed over the impeller blades and walls, the volute casing and the inlet pipe wall and the roughness of all walls is considered 100 \( \mu \text{m} \).

**RESULTS and DISCUSSION**

When a fluid with high viscosity such as heavy oil is pumped by a centrifugal pump, the performance is impaired due to increased losses in comparison to service with water. The hydraulic losses in the impeller and volute of a centrifugal pump include shock loss, diffusion loss and skin friction loss. The viscosity doesn’t influence on the shock and diffusion losses while skin friction loss is directly depend on viscosity. In the present study the influences of change in centrifugal pump geometries including change in outlet blade angle and width of impeller passage are investigated. Finally, by comparison of results, the best geometries for handling the viscous fluid is proposed.

Figure 5 shows the variation of head vs. flow rate for the six simulated geometries. The curves show that around the BEP, the head of centrifugal pump increasing in the width of passage equal to 21 mm and outlet blade angle equal to 30 degree.

![Figure 5. Comparison of H-Q diagram in simulated geometries](image)

For validating the 3D numerical simulations with experimental results, three operating performance condition are chosen for showing the best agreement in PL, BEP and OL respectively. Figure 6-8 shows the values of obtained head using experimental and 3D numerical simulation.

Comparison of the experimental and numerical magnitude of results show that the average percent error for figure 6-8 is 4%, 4.08% and 3.35% respectively. This shows the best agreement between these results.

Finally the experimental results, for three main geometries, including the variations of head, power and efficiency vs. flow rate are shown in Figure 9-11 respectively.

![Figure 6. Comparison of the experimental and numerical head vs. flow rate at 27.5 degree and 17 mm width](image)

![Figure 7. Comparison of the experimental and numerical head vs. flow rate at 32.5 degree and 17 mm width](image)

![Figure 8. Comparison of the experimental and numerical head vs. flow rate at 30 degree and 21 mm width](image)
Figure 9 shows the head variations with various flow rate, that according to these results increasing of the height of passage has more influence on the head increasing than in the outlet angle in part load (PL) performance manner. Also, in the over load (OL) performance manner, the increasing in the outlet angle and height of passage leads to decreasing in head.

The variations of power input related to the flow rate are shown in figure 10. The results illustrate that increasing in passage height leads to increasing the power input while increasing in the outlet angle leads to decreasing the power input, in a specific flow rate.

Figure 11 shows the variations of efficiency with various flow rates. It is depicted that increasing in the outlet angle in PL performance condition influences on the efficiency increasing more than increasing the height of passage.

Also the centrifugal pump performance at the 30 degree of outlet blade angle is improved in comparison with 27.5 and 32.5 degree of outlet blade angle, due to decreasing the dissipation arising by vortex formation in impeller passage when the pump handling viscose oil.

References