Numerical Investigation of the Effect of the Number of the Blades of the Impeller on the Performance of Centrifugal Pumps

Adel Pourtaghi\textsuperscript{a}, Heidar Pouladi\textsuperscript{b}

\textsuperscript{a} Space Thruster Research Institute, Iranian Space Research Center, Tabriz, Iran
\textsuperscript{b} Department of Mechanical Engineering, Islamic Azad University, Borazjan, Iran

1. Introduction

Need of human to water and replacing it from a point to another point lead to human think of making a device to resolve this problem. The first samples of pumps which driving force is satisfied by human or animals are made by ancient Egyptians in 17th century BC. Today's pumps have a variety of types which centrifuge pumps are one of them. Up to now, several studies have been done in terms of investigation and prediction of the manner of the current of fluid inside the centrifugal pumps. These studies are done for gaining characteristic curves, designing the increase of productivity and performance, interaction of fixed and movable parts of the pump, studying the manner of fluid current in designing and out of designing condition, and obtaining complex internal geometries of centrifugal pumps.

Bacharoudis and et al. [1] studied parametrically the blade of a centrifugal pump with a change of angle of outlets of impeller blades. The results indicated that there is a convenient correlation between general and local parameters. Numerical solution of three dimensional incompressible Navier-Stokes equation is impressed in an unorganized network with finite volume computational fluids' dynamic program. Flow and pressure distribution pattern in blade paths are calculated and finally, the flow rate-Headcurves are compared and discussed. Kim and et al. [2] designed optimized centrifugal pump impeller and volute using computational fluid dynamics. Barrio and et al. [3] calculated the radial load in centrifugal pumps using computational fluid dynamics. They calculated the radial load for unsteady components existing in the pump using computational fluid dynamics and then compared the results with laboratory data. Results indicated that unsteady components existing in the pump if work at out of design condition can contribute to the creation of a load equal to 40 up 70 percent of the mean load. Safa Abadi and et al. [4] in the year 2010 geometrically modeled the complex ducts of a centrifugal turbo pump using implicit and explicit measurement. Finally, they simulated permanent incompressible viscose current of amine fluid over the computational environment. In their research, also pressure fields of velocity vectors in turbopump ducts got investigated. Hong-xun Chen and et al. [5] in order to introduce gap drainage technology into practice, proposed the design concept and expression format of spatial geometry for gap drainage impeller with twisted vice blades. Salehi and et al. [6] concerned with the combined effects of geometrical and operational uncertainties on the flow field and performance of a low specific-speed centrifugal pump. The volumetric
flow rate, rotational speed and blade geometry of the pump are assumed to be stochastic with uniform Probability Distribution Functions (PDFs). Kang and et al. [7] performed an investigation on the centrifugal condensate pump which is featured by operating under cavitation condition. Both experimental and numerical techniques are utilized. Three impeller schemes are devised to improve the cavitation performance. Pump head, cavitation and flow parameter distribution are investigated systematically.

2. Identifying the equations

For this purpose, the fluid mechanic's science based on dynamic rules of Newton is used. These equations which are also called Navier Stokes Equations, in spite of enough accuracy for most of the fluid mechanics phenomenon, in many cases because of high complexity of solution are not evaluable directly and because of presence of nonlinear terms also correlation of four equations together are called as one of the most complicated equations of science world. An explicit and analytical solution of these equations is impossible, except some limited cases and considering many of simplification hypothesis [8]. The flow between two parallel plates, slow flow in a pipe with a circular cross-section and low Reynolds flow around an orb, are some examples of simple physics for which there are analytical solutions for Navier Stokes equations.

The other view which is center of attention in many of today's projects is using numerical methods. In numerical methods, first, the solution domain gets divided into very small parts named computational cells. The sum of computational cells forms a computational network. The computational cells have different shapes depending on geometrical complex and general structure of solver [9]. Computational networks are divided into two structured and unstructured types. In structured network identifying the adjacent cells is done in form of matrix and network has ordered structure. The used cells in this network are rectangles in two dimensional and hexagonal in three dimensions. This type of networks because of being ordered and special geometric requirements are not usable for many complicated geometries. In complicated geometries, unstructured networks are used. This kind of network is produced in unordered form and sometimes randomly, of course, using smart methods. The cells used in this kind of network usually are a triangle in two dimensions and tetrahedron in three dimensions. Pyramid and prism cells are used in special cases. Next, to these, the used cells in structured networks are applicable in unstructured networks too which their application is somehow less than the main cells (triangle and tetrahedron) [10].

Since in centrifugal pumps fluid is turning around the impeller, the equation must be written in two reference frames. These two sectors include fixed and rotating frames. Because of this multiple reference frame is used for expressing the equations of this system. The equations are expressed in a rotating frame and Coriolis and centrifugal forces are added to the equations as a source. The continuity and momentum equations are given below [11].

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

In above equation, the velocity component is expressed in form of relative velocity. The source term includes Coriolis and centrifugal forces based on equation (3). In this equation, the effect of gravity acceleration is neglected and it is because of the negligible height difference between inlet and outlet openings. If inlet and outlet openings have considerable height difference (like pumping suction from a deep well) the effect of gravity acceleration must be mentioned.

\[ S = -2\rho \mathbf{u} \times \mathbf{u} - \rho \mathbf{u} \times (\omega \times \mathbf{r}) \]  

In equation 3 \( \omega \) is the rotation speed of the pump.

3. Geometrical modeling

For modeling the flow of equipment related to fluids before any work it is required to model the equipment to be able to define the planes of volume or space of fluid placement. For this purpose, designed by computer knowledge is used including Solid Works, Catia, AutoCad 3D, and ... . In the present research, geometric design of pump is done by modules which are placed in Ansys Work Bench and the superiority of this work in comparison with similar cases in other softwares is that the all of the addressed points in standards is used in this software. Rotating speed, flow rate, the density of conveyed fluid, required head for the pump, the angle of entering flow, the diameter of the shaft, and diameter of the hub are parameters of designing which are determined in software. Table 1 is requirements of pump designing and figure 1 is an example of the modeled impeller in the present research.
Table 1. Requirements for designing for defining to software

| Row | Designing requirements            | Value | Unit  |
|-----|-----------------------------------|-------|-------|
| 1   | Rotating speed                    | 1450  | RPM   |
| 2   | Flow rate                         | 280   | m^3/hr|
| 3   | Conveyed fluid density            | 1000  | Kg/m^3|
| 4   | Required head for the pump        | 20    | M     |
| 5   | The angle of entering the flow    | 90    | Deg   |
| 6   | Least factor of shaft diameter    | 1.1   | -     |
| 7   | hub diameter / Shaft diameter     | 1.5   | -     |

Figure 1. Modeled impeller with 6 blades

4. Applying the boundary condition

In order to analyze the problem, the selected boundary condition for different parts of the pump is presented in table 2.

Table 2. Mentioned boundary condition

| Boundary condition application | Blows walls | Hub walls | Hub outlet wall | Inlet wall | Outlet wall | Shroud wall |
|--------------------------------|-------------|-----------|-----------------|------------|-------------|------------|
| From the conditions, Rotating and non-slip are used and the condition of the wall surface is considered rough. | From the conditions, Rotating and non-slip are used and the condition of the wall surface is considered smooth. | From the conditions, counter Rotating walls and non-slip are used and the condition of the wall surface is considered smooth. | From the conditions, fixed walls with average disturbance pattern (5% severity) are used. | Fixed walls with a flow rate of 77.8 kg/s are used. | From the conditions, counter Rotating walls and non-slip are used and the condition of the wall surface is considered smooth. |

5. Verification of analysis

For being sure of the correctness of analysis of limited element, before starting the simulation, like other simulations of limited elements the correctness of analysis must be ensured. For this purpose in the present research, the trend of simulation done by Pratap and al. [12] is repeated and we consider a parameter as common output and compute, the existing difference must be an acceptable range. In this way, it can be claimed that the done simulation is countable. The selected parameter is total pressure along the blade which is extracted in present research based on figure 2. And the condition of analysis is like table 3.
Table 3. Designing parameters

| Row | Designing parameter          | Value |
|-----|------------------------------|-------|
| 1   | Number of blades             | 5     |
| 2   | (rpm) Rotating speed         | 1450  |
| 3   | (m³/hr) Flow rate            | 280   |
| 4   | Pump head (m)                | 20    |
| 5   | The angle of inlet flow (degree) | 90   |
| 6   | Blade angle (degree)         | 22.5  |

By repeating the simulation using mentioned designing parameters and comparing total pressure distribution it can be claimed that Cantor of pressure distribution along the blade for both simulation maximum increase up to 260000 Pascal and these maximum pressures in terms of distribution almost are the same. Figure 3 illustrates the way of distribution of pressure along the blade in the present research.
The results of the centrifugal pumps analysis with an impeller having three blades

The impeller of a centrifugal pump is analyzed with three blades and without considering the volume. The performance characteristics are illustrated in table 4. The obtained head by CFD analysis is equal to 16.75 m and total efficiency is equal to 84%.

Table 4. Performance results of a centrifugal pump with three blades impeller

| Parameter                          | Value     |
|-----------------------------------|-----------|
| Rotation Speed                    | 151.8440  |
| Reference Diameter                | 0.2715    |
| Volume Flow Rate                  | 0.0780    |
| Head (IN-OUT)                     | 16.7526   |
| Shaft Power                       | 15213.4000|
| Power Coefficient                 | 0.0030    |
| Total Efficiency (IN-OUT) %       | 84.0134   |

The results of the centrifugal pumps analysis with an impeller having four blades

As it is shown in a table 5 increase in the number of blades have led to increasing of efficiency up to 89.64% and an increase of pump head up to 19.667 m.

Table 5. Performance results of a centrifugal pump with four blades impeller

| Parameter                          | Value     |
|-----------------------------------|-----------|
| Rotation Speed                    | 151.8440  |
| Reference Diameter                | 0.2715    |
| Volume Flow Rate                  | 0.0780    |
| Head (IN-OUT)                     | 19.6670   |
| Head Coefficient (IN-OUT)         | 0.1134    |
| Shaft Power                       | 16737.6000|
| Total Efficiency (IN-OUT) %       | 89.6471   |

The results of the centrifugal pumps analysis with an impeller having five blades

Head of the pump and total efficiency are from the parameters which are discussed in the present research. In impeller with five blades head of pump have increased to 21.18 meter and total efficiency has increased to 92.87%. Other parameters of pump performance of a pump with five blades impeller are shown in table 6.

Table 6. Performance results of a centrifugal pump with five blades impeller

| Parameter                          | Value     |
|-----------------------------------|-----------|
| Rotation Speed                    | 151.8440  |
| Reference Diameter                | 0.2715    |
| Volume Flow Rate                  | 0.0936    |
| Head (IN-OUT)                     | 21.1823   |
| Shaft Power                       | 20903.8000|
| Power Coefficient                 | 0.0041    |
| Total Efficiency (IN-OUT) %       | 92.7750   |

The results of the centrifugal pumps analysis with an impeller having six blades

Table 7 illustrates the performance properties resulted from 6 blades impeller pump analysis. As it can be seen, like previous states, an increase of a number of blades have led to increasing of pump head up to 22.305 m and total efficiency up to 94.366%.
Table 7. Performance results of a centrifugal pump with six blades impeller

| Parameter                          | Value       |
|-----------------------------------|-------------|
| Rotation Speed                    | 151.8440    |
| Reference Diameter                | 0.2715 [m]  |
| Volume Flow Rate                  | 0.0780 [m^3 s^-1] |
| Head (IN-OUT)                     | 22.3052 [m] |
| Shaft Power                       | 18033.7000 [W] |
| Power Coefficient                 | 0.0035      |
| Total Efficiency (IN-OUT) %       | 94.3662     |

The results of the centrifugal pumps analysis with an impeller having seven blades

As it is shown in a table 8 increase of a number of blades have led to increasing of pump head (22.702m) and total efficiency (95.098).

Table 8. Performance results of a centrifugal pump with seven blades impeller

| Parameter                          | Value       |
|-----------------------------------|-------------|
| Rotation Speed                    | 151.8440    |
| Reference Diameter                | 0.2715 [m]  |
| Volume Flow Rate                  | 0.0669 [m^3 s^-1] |
| Head (IN-OUT)                     | 22.7027 [m] |
| Shaft Power                       | 15612.4000 [W] |
| Power Coefficient                 | 0.0030      |
| Total Efficiency (IN-OUT) %       | 95.0982     |

Figure 4. The way pump efficiency has changed based on the number of impeller blades

In order to see the way number of blades affect the pump efficiency as a performance parameter the illustrated curve in figure 4 is drawn which shows the way increase of a number of blades influences the efficiency of pumps. As it is clear in curve increase of a number of blades leads to increase of efficiency of the pump. By being more careful in the way of curve growth it can be observed that slope of increase of efficiency by the change of a number of blades from three to seven decreases slowly and this shows that can select the optimum number of blades considering other designing parameter and existing problems in construction and assembly of the product.
6. Conclusions

A centrifugal pump is a pump which uses a rotating impeller for increasing the pressure of a fluid. Fluid flows to pump from the impellers hole and get accelerated by it and flows toward the volute fast from where enters to the outlet of the pump. Having enough knowledge in the field of geometrical parameters has a great effect on improving the designing and making this kind of pumps. The present study intends to take a small step in this field by evaluation of the effect of mentioned parameters and simulation of fluid current in inside the pumps. For reaching to this goal different number of blades in modeled in impellers and the performance of pump in simulated in Ansys CFX software and finally the results of CFD studies are presented as charts and curves. The results showed that by the increase of the number of blades from three to seven blades the head and efficiency of pump get increased and this value finally reaches to 22.7 and 95.1% respectively in the impeller with seven blades.

References

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

[2] J. Kim, K. Oh, K. Pyun, C. Kim, Y. Choi, J. Yoon. Design optimization of a centrifugal pump impeller and volute using computational fluid dynamics. IOP Conference Series: Earth and Environmental Science. IOP Publishing 2012. p. 032025.

[3] R. Barrio, J. Parrondo, E. Blanco. Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points. Computers & Fluids. 39 (2010) 859-70.

[4] S. Bahreinian, H. SafaAbadi. A Process to Create a Geometric Model for Fluid Conduits in Rotary Machines. Second Conference on Rotating Equipment in Oil and Power Industries 2010.

[5] H.-x. Chen, J.-w. He, C. Liu. Design and experiment of the centrifugal pump impellers with twisted inlet vice blades. Journal of Hydrodynamics, Ser B. 29 (2017) 1085-8.

[6] S. Salehi, M. Raisee, M.J. Cervantes, A. Nourbaksh. On the flow field and performance of a centrifugal pump under operational and geometrical uncertainties. Applied Mathematical Modelling. 61 (2018) 540-60.

[7] C. Kang, N. Mao, W. Zhang, Y. Gu. The influence of blade configuration on cavitation performance of a condensate pump. Annals of Nuclear Energy. 110 (2017) 789-97.

[8] A. Kakaei, P. Rahnama, A. Paykani. Numerical study of reactivity controlled compression ignition (RCCI) combustion in a heavy-duty diesel engine using 3D-CFD coupled with chemical kinetics. International Journal of Automotive Engineering. 4 (2014) 792-804.

[9] A. Nobakht, M. Shahsavani, A. Paykani. Numerical study of diodicity mechanism in different Tesla-type microvalves. Journal of applied research and technology. 11 (2013) 876-85.

[10] A. Niknam, K. Farhang. Vibration Instability in a Large Motion Bistable Compliant Mechanism due to Stribeck Friction. Journal of Vibration and Acoustics. (2018).

[11] A.-M. Georgescu, S.-C. Georgescu, C.I. Cosoiu, L. Hasegan, A. Anton, D.M. Bucur. EPANET simulation of control methods for centrifugal pumps operating under variable system demand. Procedia Engineering. 119 (2015) 1012-9.

[12] R. Pratap, S. Singh. Study of impeller design considerations of a centrifugal pump and investigate its effects on pump performance using Ansys. The journal of science, technology and humanities. (2016).