VOL. 12, NO. 8, APRIL 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

# FLOW PATTERN STUDY OF A CENTRIFUGAL PUMP USING CFD METHODS CONCENTRATING ON VOLUTE TONGUE ROLE

N. Pourmahmoud and S. Majid Taleby
Faculty of Engineering, Urmia University, Urmia, Iran
E-Mail: majid.taleby@gmail.com

## ABSTRACT

In this paper, a 3-D simulation of complex flows in a centrifugal pump (EN 80-400, Pumpiran) was performed utilizing computational fluid dynamics methods. The standard k -ε model with standard wall functions and SIMPLE algorithm were chosen for turbulence model and pressure-velocity coupling respectively. The moving reference frame was used to calculate the interaction between impeller-volute in steady condition. Also grid independency study was performed. Flow field inside impeller in the static pressure contour, path lines and velocity vector plot were shown. The head coefficients and radial force at different flow rates were predicted and they agree well with the experimental data of this pump. In all simulation results the effect of volute tongue on the flow field was described. Finally the interpretations of results indicated that for efficiency enhancement, volute requires to redesign.

**Keywords:** computational fluid dynamics, centrifugal pump, MRF method, CFD, volute tongue.

## INTRODUCTION

Centrifugal pumps are used in a widespread range of applications to enhance energy content of a liquid flowing through them. They transforms energy of an initial mover (an electric motor or turbine) first into velocity or kinetic energy, so into pressure energy of a pumped liquid. Centrifugal pumps have two main parts: impeller and volute or diffuser. The impeller is the rotating part that transforms driver energy into the kinetic energy. The volute or diffuser is the stationary part that converts the kinetic energy of the liquid into pressure energy.

Due to the extensive complication of flow in the centrifugal pumps, because of the three dimensional developed structures, turbulence, secondary flows, unsteadiness, etc., there exist still many unknown issues relevant to the complete flow field in these pumps, which require to be studied. Also the conduction of experimental investigations on models with various volute and impeller geometries is time-consuming and costly, and due to the complicated geometry, it is not feasible to accomplish a thorough study of the flow pattern for a vast number of operating conditions. Therefore, Computational fluid dynamics (CFD) has recently become a suitable method of study of the flow patterns and losses.

Computational fluid dynamics (CFD) have successfully accomplished the prediction of the flow through the pumps and the improvement of their design. Various researchers have considerably contributed to evaluating the flow field inside centrifugal pumps with considering the interaction between impeller and volute or vane diffuser, in order to reach the design of high performance centrifugal turbo machines. For instance, Shojaeefard *et al.* [1, 2], Asuaje *et al.* [3], Majidi [4], Huang *et al.* [5] and Zang *et al.* [6] investigated flow field inside centrifugal pump and performed parametric study utilizing computational fluid dynamics methods. Nevertheless necessity to the serious works that result in correction of the existence design or new design of centrifugal pumps is severely sensible.

In this article, the procedure of 3-D investigation of flow in centrifugal pumps includes the sections of geometry definition, mesh generation, mesh independency study, boundary condition, solver formulation, and processing of results. The obtained numerical results are compared with the experimental ones, and acceptable correlation is found between the two sets of results. The flow field inside pump was investigated and concentrated on the volute tongue role in the performance of pump.

## MODEL DEVELOPMENT

In this study, a commercial centrifugal pump (EN 100-400 manufactured by PUMPIRAN) was investigated. The main pump parameters are presented in Table-1.

**Table-1.** Geometrical parameters of the pump.

| Pump EN 80-400 (PUMPIRAN) | |
|---|---|
| **Parameter** | **Value** |
| **Impeller** | |
| Inlet flange diameter | 100 mm |
| Impeller inlet diameter | 122 mm |
| Impeller outlet diameter | 404 mm |
| Impeller width | 12 mm |
| Outlet blade angle | 28° |
| Blade number | 7 |
| **Volute** | |
| Base volute Diameter | 420 mm |
| Volute width | 21mm |
| Outlet flange diameter | 60 mm |
| **Operational Condition** | |
| Angular Velocity | 1450 rpm |
| Flow rate at BEP | 92.5 $m^3$/hr. |
| Head at BEP | 51 m |
| Specific Speed | 12.2 |

## NUMERICAL SIMULATION

### Grid generation

The geometry of pump is divided into two zones: rotational and stationary. Surface between impeller (moving zone)-volute (stationary zone) defines the grid as interface. For complex geometry like a pump, unstructured way mainly with tetrahedral elements is used for grid generation, but other cell forms like pyramids, hexahedra and wedges also occur.

For boundary layer simulation and concentrating near the wall to maintain a value of y+ less than 10 [7], some prism layers with controlled height is added near the blades grid. A view of the generated grid can be seen in Figure-1, while a detail of the impeller grid is shown in Figure-2.

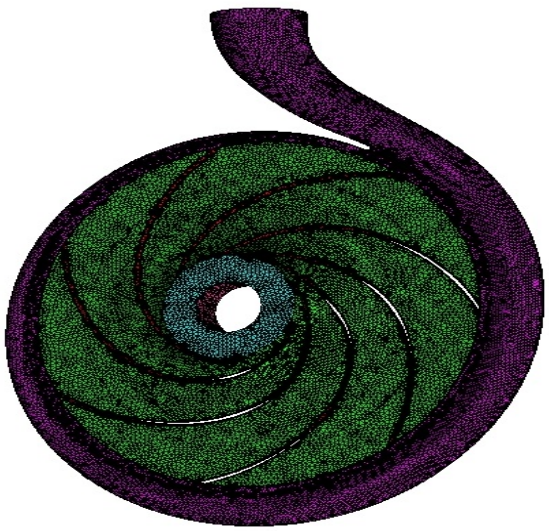

**Figure-1.** An overview of grid domain.

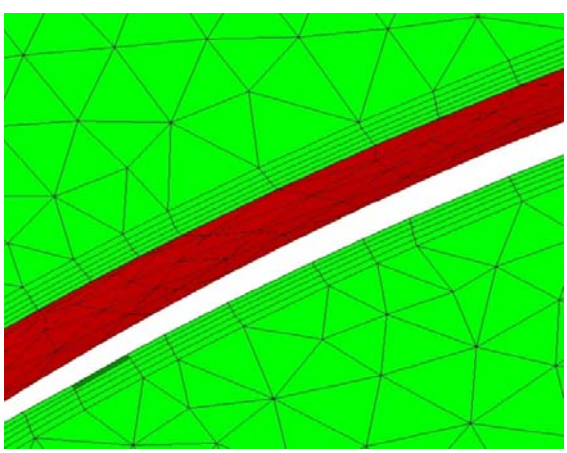

**Figure-2.** Prism layers in neighborhood of blades.

Generally, in a pump it is calculated that the skewness factor of the grid should exceed 0.80 [8]. As the model generated is assumed to have a skewness factor generally in the order of 0.80, there is no problem with the grid quality.

Theoretically, the errors in the solution related to the grid must disappear for an increasingly fine mesh [9]. The flow rate at nominal flow conditions was taken as the parameter to evaluate four grids (Table-2) and determine the influence of the mesh size on the solution.

**Table-2.** Grid sizes.

| Characterstic | Number of Elements |
|---|---|
| Grid A | 2,612,000 |
| Grid B | 4,179,000 |
| Grid C | 7,320,000 |
| Grid D | 8,502,000 |

The selected convergence criteria were a maximum residual of $10^{-5}$. In Figure-3, it is observed how the calculated flow rate reaches an asymptotic value as the number of nodes increases. According to this figure, grid B (4179281 elements) is considered to be sufficiently reliable to ensure mesh independence.

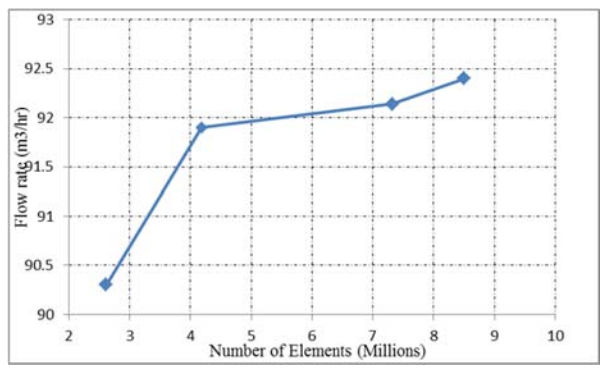


**Figure-3.** Influence of grid size on flow rate.

### Boundary condition

In order to simulation of real condition of pump performance test, in the present study, pressure inlet and pressure outlet boundary conditions were used for the suction and discharge, respectively. Over the walls involving hub, shroud, blades and volute casing, no-slip boundary conditions have been imposed, and the roughness of all walls is considered 0.1 mm. Turbulent conditions at suction and discharge are defined bythehydraulic diameters of inlet and outlet and 1% turbulence intensity.

The fluid used in the numerical simulation and experimental investigation, is water with the density of 998 kg/m$^3$ and kinematic viscosity of 1 mm$^2$/s at 20º C.

### Solver formulation

In the present study, segregated solution algorithm with a control volume base technique is utilized. The segregated solution is preferred, due to the benefits over the alternative technique of strong coupling between the velocities and pressure. This can help to prevent

convergence problems and oscillations in pressure and velocity fields. In this technique, governing equations of mass, momentum and turbulence on the individual cells within the computational domain are integrated to create algebraic equations for each unknown dependent variable. The pressure and velocity are couple dusing the semi-implicit method for pressure linked equations (SIMPLE) algorithm [10, 11] which uses a guess-and-correct procedure for the calculation of pressure on the staggered grid arrangement. The second order upwind interpolation scheme is employed for the discretization of the governing equations. The first/second-order upwind scheme is always bounded and provides stability for the discretization of the pressure-correction equation.

The standard k -ε model is activated to handle the turbulence effects. The standard wall function is chosen to estimate wall shear stress and pressure more precisely.

To simultaneously simulate flow through rotational and stationary zone multiple reference frame (MRF) model is utilized. MRF is a steady-state approximation and is appropriate if the flow at the boundary between the stationary and moving regions will be nearly uniform.

## SIMULATION RESULTS AND DISCUSSION

### Velocity fields

Figure-4 shows the path lines of the fluid in the orthogonal plane at the middle of the impeller and the volute for different flow rates. This figure indicates some important and noticeable points: first at the nominal flow, pathlines are in direction of blades, in other words the flow in the impeller is blade-congruent, and there isn't any circulation zone inside the impeller, which confirms the proper design of impeller at the nominal flow. Second the circulation zone that appears after the volute tongue grows with increasing flow rate. It should be considered that in the optimum volute design this zone should be minimum at the nominal flow. At last the circulation zones at low flow rates appear inside the impeller resulting decreasing of hydraulic efficiency.

Figure-5 represents absolute velocity vectors infront of the volute tongue. The spots show the approximate location of the stagnation point. According to publish data [12], the stagnation point at nominal flow should be place in the middle of the tongue edge. But in this pump in at the nominal flow, stagnation point is located in the internal side of tongue; which expresses the position of volute tongue is improper.

### Pressure distribution inside the pump

Figure-6 Shows static pressure distribution in the orthogonal plane at the middle of the impeller and volute at different flow rates. Due to work of impeller, static pressure increases with the flow route in the impeller. In spiral of the volute, static pressure keeps up while kinetic energy turns into potential energy and reaches maximum value till some location. Afterwards, pressure value decreases in some degree due to hydraulic loss on the way to the exit of the pump.

It can be seen that the contours are almost circular curves, which shows very small proportion of static pressure rise brought is due to the reduction of relative velocity. Furthermore the pressure of corresponding points on the pressure side is approximately equal to the suction side so the fluid in impeller channels will not experience any circulation and loss of energy.

According to the texts [13] for flow rates lower than nominal, the pressure reaches its lowest value directly in front of the tongue and starts to increase around the periphery of the impeller, taking its maximum value just behind the tongue. When the flow rate approaches the nominal, the pressure distribution is smoothed achieving the most uniform contour in nominal volume flow rate. At higher flow rates, the pressure decreases gradually from its maximum value in front of the tongue to a minimum value just behind the tongue. The aforementioned verify remarkable influence of the tongue in the pressure field inside the pump. In the Figure-6, it is clear that pressure distribution reaches smooth condition at 0.78 nominal flow rate and at the nominal flow rate, the contour is similar to the higher flow rates contour. This again indicates the improper design of volute for the specified nominal flow.

The non-uniform static pressure field around the impeller results in a radial force which is calculated by the integration of the pressure force distribution. In Figure-7 the radial force coefficients at different flow rates are shown. For volume flow rates close to the nominal, a more uniform pressure distribution exists which implies less radial forces, so it is expected that the minimum radial force is seen in the nominal flow rate. But in this pump due to the reasons described previously, minimum radial force is seen in the 0.78 nominal flow rate.

### Performance curve

Figure-8 shows the head coefficient (ψ) as a function of flow rate ratio (Q/QN) using: a) one dimensional analysis data (Theoretical), b) experimental data, c) CFD simulation results, d) CFD simulation results that ignored the volute losses and just simulated impeller. Theoretical curve is calculated utilizing Equation.

(1)[14]:

$$\psi_{th} = 2\left(1 - \tau_2 \frac{\varphi_2}{\tan \beta_{2B}}\right) \tag{1}$$

Where Ψ is head coefficient and defined as Equation. (2), g is gravity acceleration, h is the total head, u is the circumferential velocity at blade tip, $\tau_2$ is the blockage factor, $\varphi_2$ is the flow coefficient and $\beta_{2B}$ is the blade angle.

$$\psi = \frac{2 \times g \times h}{u^2} \tag{2}$$

Comparing experimental curve with CFD curve indicates that MRF technique is capable in predicting of performance curve. In low flow rates prediction has good precision, conversely in high flow rate simulation precision decreases and the calculations give a serious under-prediction of the head.

In the impeller (CFD) curve the high quantities of head coefficient relative to the pump overall simulation results represents again the incorrect volute design.

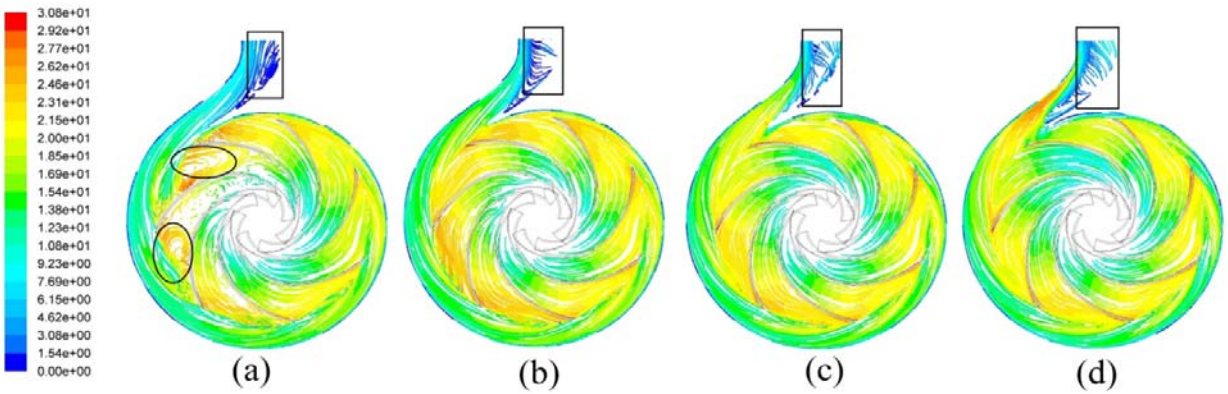


**Figure-4.** Path lines of the fluid in the orthogonal plane at four flow rates colored with velocity magnitude (m/s):(a)Q/QN=0.53, (b) Q/QN=0.78, (c) Q/QN=1.0, (d) Q/QN=1.28(Circulation zones are indicated).

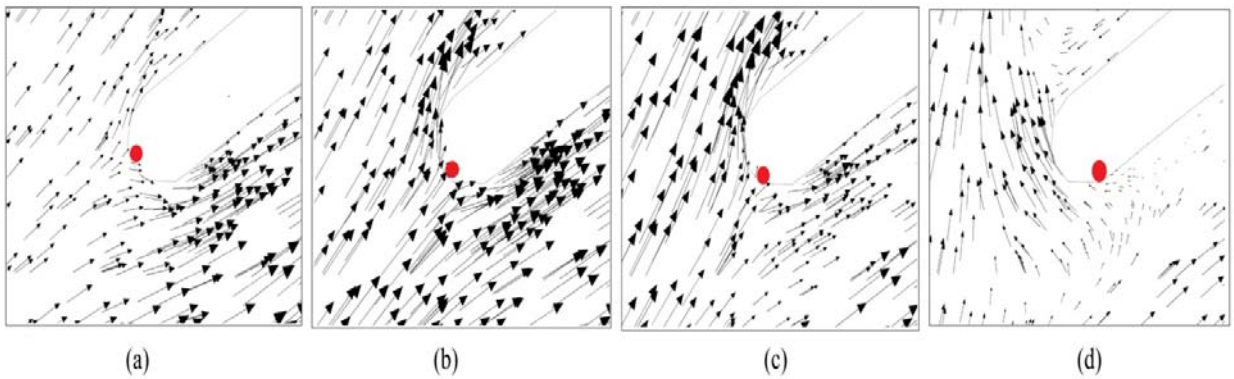


**Figure-5.** Velocity vectors in the region close the tongue at four flow rates: (a) Q/QN=0.53, (b) Q/QN=0.78, (c) Q/QN=1.0, (d) Q/QN=1.28 (Spots represents approximate location of stagnation point).

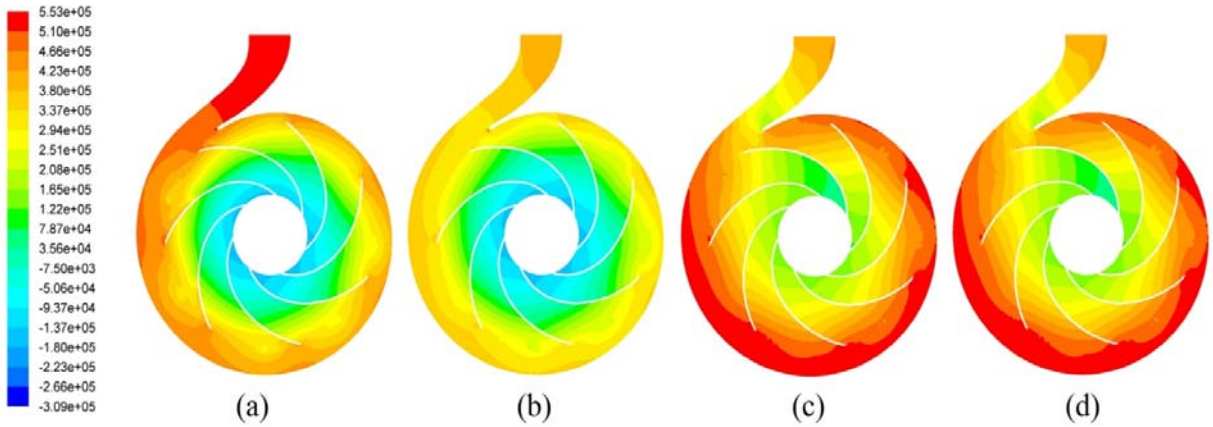


**Figure-6.** Contours of static pressure at four flow rates: (a) Q/QN=0.53, (b) Q/QN=0.78, (c) Q/QN=1.0, (d) Q/QN=1.28.

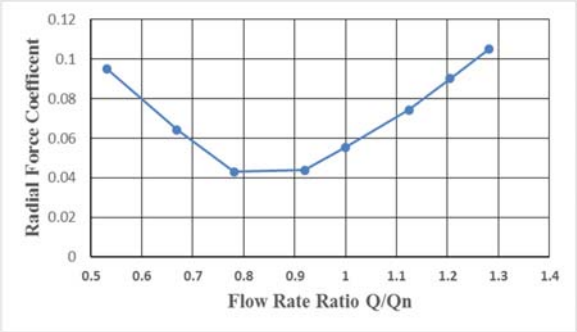


**Figure-7.** Radial coefficient versus flow rate ratio.

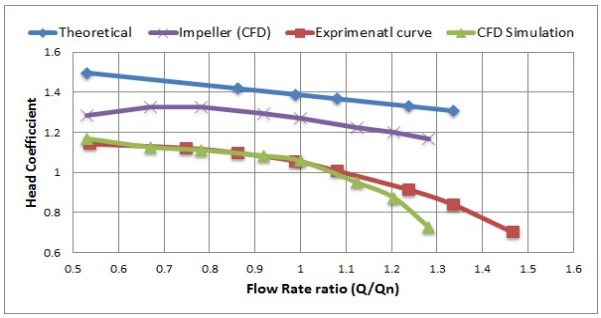


**Figure-8.** Head coefficient versus flow rate ratio curve.

## CONCLUSIONS

The main conclusions of this study are below:

- Comparing between experimental head coefficient and CFD curve represents that CFD simulation utilizing MRF method is capable to predict of pump performance with acceptable error.
- Comparing between impeller head coefficient and theoretical curve represents the proper design of impeller.
- Whereas static pressure contour, pathlines, velocity vectors plot and radial force curve, all indicate that the improper design of volute. In result for the efficiency enhancement, the volute redesign is imperative affair.

## ACKNOWLEDGEMENTS

The authors acknowledge the support from the Pumpiran Company. All numerical computations have been performed at the R&D department of the Pumpiran Company.

## REFRENCES

[1] M. H. Shojaeefard, F. A. Boyaghchi and M. B. Ehghaghi.2006.Experimental study and three-dimensional numerical flow simulation in a centrifugal pump when handling viscous fluids. IUST International Journal of Engineering Science. 17(3-4): 53-60.

[2] M. H. Shojaeefard, F. A. Boyaghchi and M. B. Ehghaghi. 2007. Studies on the influence of various blade outlet angles in a centrifugal pump when handling viscous. American Journal of Applied Sciences. 4(9): 718-724.

[3] M. Asuaje, F. Bakir, S. Kouidri, F. Kenyery and R. Rey. 2005. Numerical modelization of the flow in centrifugal pump: volute influence in velocity and pressure fields. International Journal of Rotating Machinery. 3: 244-255.

[4] K. Majidi. 2005. Numerical study of unsteady flow in a centrifugal pump. ASME Journal of Turbomachinary. 127: 363-371.

[5] Si. Huang and Y. L. Wu. 2006. Analysis of flow field asymmetry and force on a centrifugal pump by 3-D numerical simulation. Journal of Fluid Machinary. 2(008).

[6] M. J. Zhang, C. G. Gu and Y. M. Miao. 1994. Numerical study of the internal flow field of a centrifugal impeller. ASME paper 94-GT-357.

[7] J. H. Kim, Y. S. Choi and P. J. Park. 2006. Journal of Fluid Machinary. 9(2):41-42.

[8] K. M. Guleren and A. Pinarbasi. 2004. Numerical simulation of the stalled flow within a vaned centrifugal pump. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science. 218(4): 425-435.

[9] J. H. Ferziger and M. Peric. 2002. Computational methods for fluid dynamics Vol. 3.Springer Inc. Berlin.

[10] S. Derakhshan and A. Nourbakhsh. 2008. Theoretical, numerical and experimental investigation of centrifugal pumps in reverse operation. Experimental Thermal and Fluid Science. 32(8): 1620-1627.

[11] K. W. Cheah, T. S. Lee, S. H. Winoto and Z. M. Zhao. 2007. Numerical flow simulation in a centrifugal pump at design and off-design conditions. International Journal of Rotating Machinery. 8. Article ID 83641.

[12] J. Gonzalez, J. Fernandez, E. Blanco and C. Santolaria. 2006. Steady and unsteady radial forces for a centrifugal pump with impeller to tongue gap variation. Journal of Fluids Engineering. 128(3): 452-462.

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

[14] J. F. Gülich. 2008. Centrifugal pumps. Springer Inc. Berlin.