Validation Of Hydraulic Design Of A Metallic Volute Centrifugal Pump

Rakesh Sidhesware [1*], Dr. O D Hebbal [2*]

[1*]*PG Student, Thermal Power Engineering, PDA College of Engineering, Gulbarga-585102, Karnataka (INDIA)*
[2*]*Professor, Department of Mechanical Engineering, PDA College of Engineering, Gulbarga-585102, Karnataka (INDIA)*

Abstract

Centrifugal pump is a most common pump used in industries, agriculture and domestic applications. The design of a centrifugal pump impeller demands a detailed understanding of the internal flow during design and off-design operating conditions. The present paper describes the simulation of the flow in a centrifugal pump impeller at four different flow rates viz. 3.33, 7.91, 12.52 and17.06 kg/s, with working fluids as Petrol/ Diesel and VG-32.The numerical solution of the deiscretized three-dimensional, incompressible Navier-Stokes equations over an unstructured grid is accomplished with CFD package ANSYS-CFX. For each flow rates, performance results, blade loading plots, mass averaged total pressure and static pressure, area averaged absolute velocity are presented.

Keywords: Centrifugal pump impeller, CFD analysis, Flow rates, Blade loading, Velocity vectors.

1. Introduction

The complexity of the flow in any turbomachine is due to the three dimensional blade geometry, turbulence, secondary flows, unsteadiness etc. Computational fluid dynamics (CFD) has successfully contributed to the prediction of the flow through pumps and the enhancement of their design. Various researchers have considerably contributed to reveal the flow mechanisms inside centrifugal pump impellers with spiral volute or vaned diffuser volute aiming to the design of high performance centrifugal
turbomachines. Computational analysis of a centrifugal pump presented by Bcharoudis et al., 2008 reveals the flow mechanisms inside centrifugal impellers and performance by varying outlet blade angle. They observed a gain in head more than 6% with increase in outlet blade angle from 20o to 50o.Computational analysis of a centrifugal pump for off-design volume flow rate is done by Khin Cho Thin et al., 2008. They predicted the performance and calculated impeller volute disc friction loss, slip, shock losses, recirculation losses and other frictional losses. Optimization of blade inlet geometry of centrifugal pump impeller is done by Vasilios et al., 2005.
John S. Anagnostopoulos, 2006 solved RANS equations for the impeller of centrifugal pump and developed fully automated algorithm for impeller design. Computational analysis on a centrifugal
pump handling viscous fluids is presented by Shojaee Fard and Boyaghchi, 2007. They observed performance improvements in centrifugal pump with increase in the outlet blade angle due to decrease of wake formation at the exit of the impeller.
Experimental investigation of centrifugal pump impeller handling both water and viscous oil is done by Wen-Guang, 2006. He observed that the blade discharge angle has significant influence on the head, shaft power and efficiency of the centrifugal oil pump at various viscosity conditions. The effects of flow behavior in a nontraditional
Centrifugal pump, whose diffuser was subjected to three different radial gaps (10%, 15%, and 20%), were investigated numerically by Adnan Ozturk et al.,2009.
The last decade research involves more advanced computational results. Recently, Milan Sedlr et al., 2009, Min-Guan Yang et al., 2007, Timar, 2005, Barrio et al., 2010, TAN Minggao et al., 2010, and Cheah et al., 2007, extended the prediction of the performance at various operating
Conditions. Several algorithms have been proposed and developed, targeting to the numerical simulation of the flow field of a centrifugal pump impeller. These algorithms apply either pressure based or density based methods for the solution of Navier Stokes equations.

2. Pump Specifications

The specification of centrifugal pump undertaken in the current analysis is shown in Table No. 1. The centrifugal pump impeller model with six blades is shown in Fig.1.

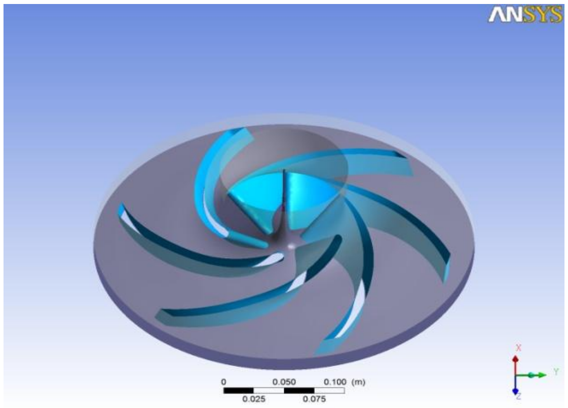


Fig.1. 3D Model of the radial impeller.

Table No. 1: Specifications of Centrifugal Pump Impeller

Blade width b	20mm
Diameter of the impeller at the suction side D1	150mm
Diameter of the impeller at the pressure side D2	280mm
Outlet blade angle 2	20o
Pump head H	10 m
Speed of the impeller N	925 rpm
Flow rate, Q	0.0125 m3/sec
The diameter of impeller eye D0	43.5mm
Hydraulic Efficiency	83%
Number of Blades	06

3. Governing Equations:

The incompressible flow through the rotating impeller is solved with a moving frame of reference with constant rotational speed. 3-D incompressible Navier-Stokes equations with the rotational force source term are solved to analyze the flow in centrifugal pump. Turbulence is modeled with k-ε turbulent model.

3.1 Mass conservation equation:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_j}(\rho U_j)=0 \ldots \qquad (1)$$

In this equation Uj represents the three dimensional velocity vector components of the flow.If the flow is assumed steady, $\frac{\partial \rho}{\partial t}=0$ and the equation reduces to

$$\frac{\partial U_j}{\partial x_j}=0 \cdots \qquad (2)$$

3.2 Momentum conservation equation:

The conservation equation for momentum, Ui, can be formulated as

$$\frac{\partial}{\partial t}(\rho U_i)+\frac{\partial}{\partial x_j}(\rho U_i U_j)=-\frac{\partial P}{\partial x_i}-\frac{\partial \tau_{ij}}{\partial x_j}+\rho f_i \ldots \qquad (3)$$

The three terms on right hand side of above equation represent the xi components of all forces due to the pressure, P, the viscous stress tensor, ij, and the body force, fi. For a Newtonian fluid, the stress tensor is given by

$$\tau_{ij}=-\mu_b \delta_{ij}\left(\frac{\partial U_i}{\partial x_i}+\frac{\partial U_j}{\partial x_j}+\frac{\partial U_k}{\partial x_k}\right)-\mu\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\ldots \qquad (4)$$

whereμ b = μ2/3 is the bulk viscosity,μ is the dynamic viscosity and δ ij represents the Kronecker delta (δ ij = 1 for i=j and δij = 0 for i j). For flows in rotating frames of reference, the effects of the Coriolis and centripetal forces are modeled in the code. In this case

$$\vec{f}_i=-\ 2\vec{\Omega}\times\vec{U}+\vec{\Omega}\times(\vec{\Omega}\times\vec{r})\ \ldots \qquad (5)$$

Where vector notation has been used:x is a cross product is the rotation velocity and r is the location vector.

3.3 Energy conservation equation

Besides mass and momentum, energy is third fluid property for which a conservation equation in terms of the total enthalpy, H is given by

$$\frac{\partial}{\partial t}(\rho H)+\frac{\partial}{\partial x_j}(\rho U_j H)=\frac{\partial P}{\partial t}-\frac{\partial}{\partial x_j}(U_i \tau_{ij}+Q_j)+\rho U_i f_i \ldots \qquad (6)$$

where ,H=h+0.5 UiUi, h=Static enthalpy
if dissipation is small, neglecting pressure and dissipation terms

$$\frac{\partial}{\partial t}(\rho H)+\frac{\partial}{\partial x_j}(\rho U_j H)=-\frac{\partial Q_j}{\partial x_j}+\rho U_i f_i \ \ldots \qquad (7)$$

In a rotating frame of reference, the rothalpy, I, is advected in place of the total enthalpy, H.
The equation for I is given by

$$I=H-\frac{\omega^2 R^2}{2}\ldots \qquad (8)$$

where the rotation rate and R is the local radius.

4. Meshing

The geometry and the mesh of a six bladed Pump impeller domain were generated with ANSYS Workbench. Unstructured meshes with tetrahedral cells are used for the domain of impeller as shown in Fig.2.

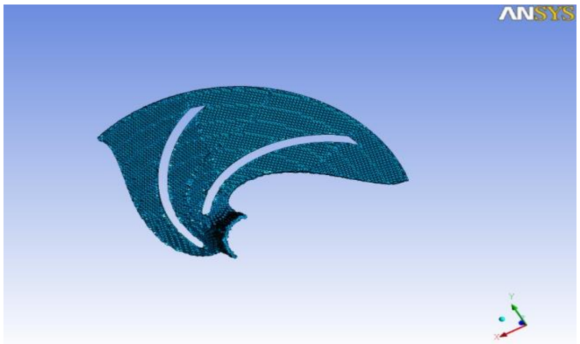


Fig.2. Unstructured mesh of the impeller

The mesh is refined in the near tongue region of the volute as well as in the regions close to the leading and trailing edge of the blades. Around the blades, structured prismatic cells are generated to obtain better boundary layer details. Fig.3 shows the mesh refinement around the blade surface and inflated layers around the blades. A total of 3,570,268 elements are generated for the impeller domain. Mesh statistics are presented in Table No.2

Table No .2. Mesh Statistics

Total number of nodes	746141
Total number of tetrahedral	3318311
Total number of pyramids	879
Total number of prisms	251078
Total number of elements	3570268

5. Boundary Conditions

Centrifugal pump impeller closed type domain is considered as a rotating frame of reference with a rotational speed of 925 rpm anti clockwise direction. The working fluid through the pump is water at 25oC. Casing is not taken into the account for analysis. k- Turbulence model with turbulence intensity of 2% is considered. Non slip boundary conditions have been imposed over the impeller blades, hub and shroud. Roughness of all walls is considered 100"m. Inlet static pressure and outlet mass flow rate are given based on flow coefficients. Convergence precision of residuals was considered as 10-5. Three dimensional incompressible N-S equations are solved with ANSYS-CFX Solver.

6. Result

Centrifugal pump impeller is solved for four different flow rates. The performance results for different working fluids are shown in respective tables:

Table no: 3. performance results of petrol as working fluid.

Mass flow rate (kg/s)	3.33	7.91	12.52	17.06
Head coefficient	0.2146	0.1929	0.1246	0.1600
Head (LE-TE) (m)	15.90	14.20	12.90	11.86
Head (IN-OUT) (m)	16.05	14.43	13.06	11.97
Shaft power (W)	574.52	1221.19	1741.04	2192.3
Static efficiency(%)	58.45	60.86	64.13	65.33

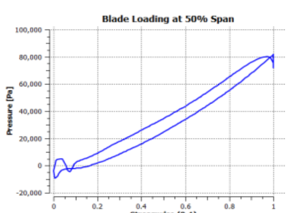
Table no: 4. performance results of Diesel as working fluid.

Mass flow rate (kg/s)	3.33	7.91	12.52	17.06
Head coefficient	0.2162	0.1926	0.1699	0.1543
Head (LE-TE) (m)	15.89	13.91	12.49	11.39
Head (IN-OUT) (m)	16.19	14.42	12.72	11.55
Shaft power (W)	585.70	1241.0	1742.03	2189.82
Static efficiency(%)	56.97	59.03	62.40	62.80

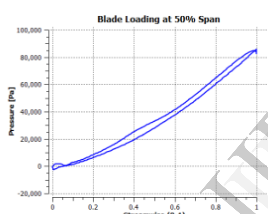
Table no: 5. performance results of VG-32 as working fluid.

Mass flow rate (kg/s)	3.33	7.91	12.52	17.06
Head coefficient	0.2145	0.1919	0.1740	0.1594
Head (LE-TE) (m)	15.83	14.11	12.87	11.83
Head (IN-OUT) (m)	16.07	14.37	13.03	11.93
Shaft power (W)	574.00	1220.23	1736.62	2187.96
Static efficiency(%)	58.48	60.84	64.14	65.23

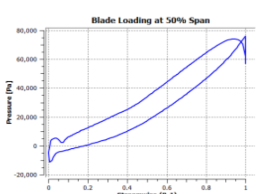
6.1 Blade loading at 50% span location for different flow rates:



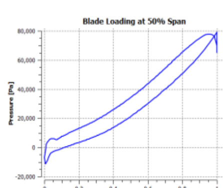
Q = 3.33 kg/s



Q = 7.91kg/s



Q = 12.52 kg/s



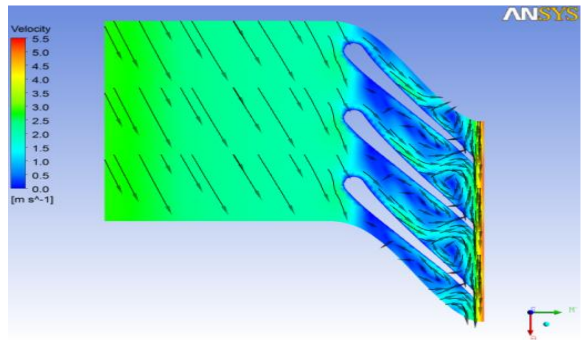
Q = 17.91 kg/s

Fig .3.Blade loading at 50% span location for different flow rates

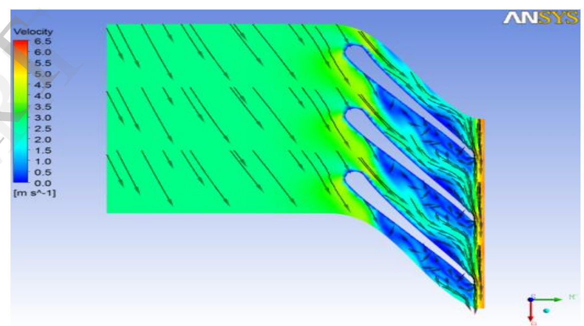
Above fig. shows the Blade loading at 50% span location for different flow rates. High pressures on pressure side of the blade and low pressures on suction side of the blade are observed as shown in Fig. 4.1.7. When the flow rate is increasing, at leading edge, there is a pressure drop on pressure side is observed. This is due to a weak vortex at the pressure side of the blade.

6.1 Velocity vectors at the mid span for different flow rates:

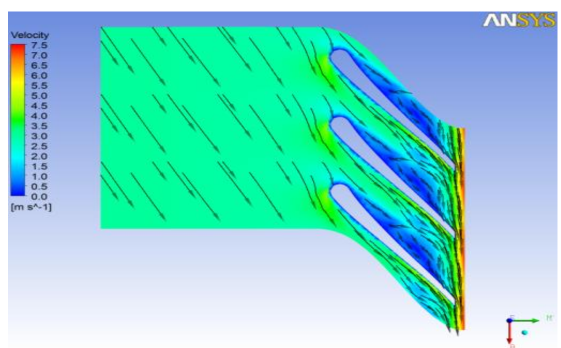
At all the flow rates, the fluid flows smoothly along the blade walls. The blade curvature exhibits a weak vortex at the pressure side of the blade. Below fig shows the low pressure regions. It is observed that the absolute velocity vector distribution is not uniform in the blade passage across the impeller width.



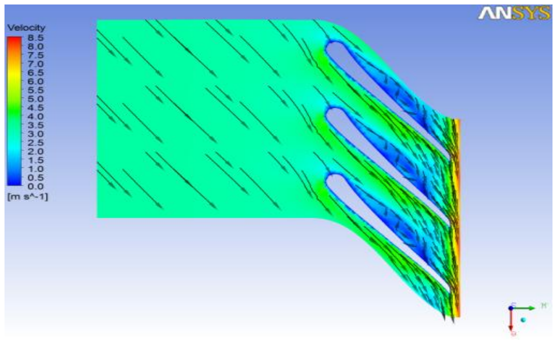
Q = 3.33kg/s



Q = 7.91 kg/s



Q = 12.52 kg/s



Q = 17.06 kg/s

Fig.5.Velocity vectors at the mid span for different flow rates.

7. Graphs:

7.1 Head v/s flow rates

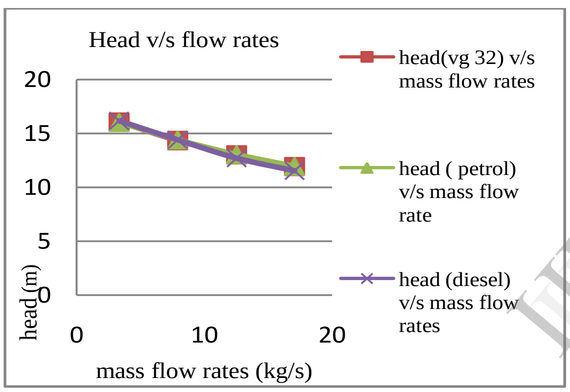


Fig.6 shows variation of centrifugal pump head with increase in discharge.

From the fig centrifugal pump head of different fluids used as working fluid follows the same pattern as in case of water. As mentioned, the speed of the pump was kept constant. It can be observed that as discharge increase, head decreases. However, the nature of head versus discharge curve is similar to that of standard pump curve.

6.2 Efficiency v/s flow rates:

From the fig.7 It shows that as mass flow rates increases the efficiency also increases. It has same pattern as in water (working fluid).But in case of diesel the performance of the centrifugal pump is lower than the other working fluids. As mentioned, the speed of the pump was kept constant. It can be observed that as discharge increase, the efficiency increase, reaches maximum at rated condition and then decreases when discharge increase beyond rated conditions, i.e. parabolic profile. It was observed that the maximum efficiency of the pump is achieved at the discharge of 12.52kg/s as per the paper results as well as CFD analysis.

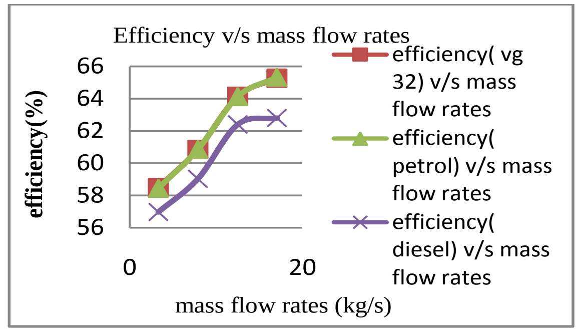


Fig.7.shows variation of pump efficiency with increase in discharge

7.3 Power v/s flow rates:

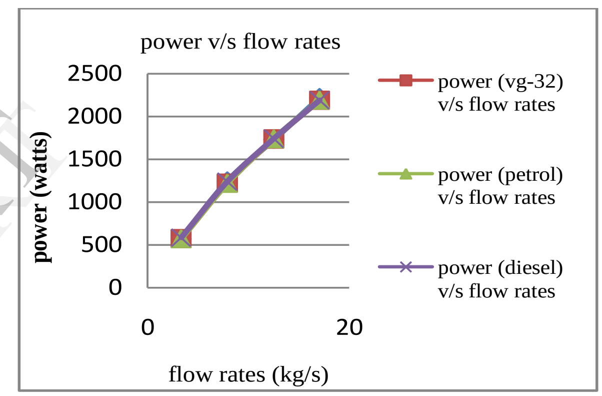


Fig.8. shows variation of input power at the pump shaft with increase in discharge.

From the graph, it shows as the flow rates of the centrifugal pump increases the power also increases and it follows for all the working fluids. And it doesn't effect in performance of the centrifugal pump. To plot this curve, the speed of the pump was kept constant and variation of shaft power input was plotted corresponds to increase in discharge. It can be seen that as discharge increase the power input for the pump increase. Power predicted by CFD analysis is higher than that of paper results. However, the nature of power versus discharge curve is similar to that of standard pump curve.

8. Conclusion:

A centrifugal pump impeller was designed and analyzed with the aid of computational flow dynamics. The increase of the designed flow rate causes a reduction in the total head of the pump. With the increase of mass flow rates drop in static

ISSN: 2278-0181
Vol. 2 Issue 7, July - 2013

pressures are observed from pressure contours on mid span for different flow coefficients.

At designed or more than designed mass flow rate, the fluid flows smoothly along the blade walls. The blade curvature exhibits a weak vortex at the pressure side of the blade. On pressure side of the blade static pressure drop is observed. At low mass flow rates a recirculation zone is established near the leading edge of each blade.

With regards to the computational simulation in the present study the following conclusions can be drawn

- There is a decrease in head by increasing the flow rates. CFD results are closely matching with the experimental results.
- Pressure distribution over the suction and pressure side of the blades when flow rate is nominal is clearly appreciable.
- Head and the efficiency of the centrifugal pump decreases with the increasing flow rate.
- From pressure contours on mid span for different flow rates the pressure increases higher values at the lowest flow rate and the pressure are decreasing with increasing flow rates.

9. Referance:

[1.] E.C. Bcharoudis, A.E. ilios ,M.D. Mentzos and D.P. Margaris, ”Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle”, The open Mechanical Engineering Journal, pp75-83, 2008,.

[2]. Khin Cho Thin, Mya Mya Khaing, and Khin Maung Aye, ”Design and Performance Analysis of Centrifugal Pump”, World Academy of Science, Engineering and Technology pp46,2008.

[3.] Vasilios A. Grapsas, John S. Anagnostopoulos and Dimitrios E. Papantonis, “Hydrodynamic Design of Radial Flow Pump Impeller by Surface Parameterization”, International Conference on Experiments / Process / System Modeling / Simulation / Optimization, Athens, 6-9 July, 2005.

[4]. John S. Anagnostopoulos, “CFD Analysis and Design Effects in a Radial Pump Impeller”, Wseas Transactions on Fluid Mechanics, Vol 1, July2006.

[5]. M.H.Shojaee Fard and F.A.Boyaghchi, “Studies on the influence of various blade outlet angles in a centrifugal pump when handling viscous fluids”, American Journal of Applied Sciences pp.718-724, 2007.

[6] Motohiko Nohmi and Akira Goto, “Cavitation CFD in a Centrifugal pump”, Fifth international symposium on cavitations, Osaka, Japan, november2003.

[7]. Wen-Guanf LI, “Blade Exit Angle Effects on Performcnace of a Standard Industrial Centrifugal Oil Pump”, Department of Fluid Machinery, Lanzhou University of Technology, Lanzhou, P R China.

[8]. Adnan Ozturk, Kadir Aydin, Besir Sahin and Ali Pinarbasi, “Effect of impeller –diffuser radial gap ratio in a centrifugal pump”, Journal of Scientific & Industrial Research, Vol. 68.March 2009,pp.203-213.

[9]. G. Pavesi , “Impeller Volute and Diffuser Interaction”, Department of Mechanical Engineering,University of Padova,Via Venezia,ITALY, RTO-EN-AVT-143,Vol.124,June 2002., pp.348-355.

[10]. Byeong Rog Shin and Byeong Rog Shin,” Investigation of Suction Vortices Behavior in Centrifugal Pump”, Japan-Korea CFD Workshop 2009.

11. Milan Sedlr, Jiri Soukal, and Martin Komarek,” CFD Analysis of middle stage of multistage pump operating in Turbine regime”, Journal of Engineering MECHANICS, Vol. 16, 2009, No. 6,pp. 413–421.

12. Min-Guan Yang, Dong Liu and Xiang Dong,” Analysis of Turbulent flow in the impeller of a chemical pump”, Journal of Engineering Science and Technology, Vol. 2, No. 3 (2007), 218 – 225.

13. P. TIMÁR,” Dimensionless Characteristics of Centrifugal Pump”, Department of Chemical and Biochemical Engineering, Faculty of Chemical and Food Technology, Slovak University of Technology, April 2005, pp812-837.

14. R. Barrio, J. Fernández, J. Parrondo and E. Blanco, “ Performance prediction of a centrifugal pump working in direct and reverse mode using Computational Fluid Dynamics”, International Conference on Renewable Energies and Power Quality,(ICREPQ‟10),Granada (Spain), 23rd to 25th March, 2010.

15. TAN Minggao, YUAN Shouqi, LIU Houlin, WANG Yong, and WANG Kai, “Numerical Research on Performance Prediction for Centrifugal Pumps”, chinese journal of mechanical engineering, vol. 23, 2010.