Investigation of CFD Based Flow Analysis of Centrifugal Pump

T. Vinod Kumar[1], P. Vivek[2], S.Arunkumar[3], A.Arul peter[4]
[1,3]Assistant Professor, Department of Mechanical Engineering, Vels Institute of Science, Technology & Advanced Studies, Chennai, India
[2]Research Scholar, Department of Mechanical Engineering, Vels Institute of Science, Technology & Advanced Studies, Chennai, India
[4]Associate Professor, Department of Mechanical Engineering, Vels Institute of Science, Technology & Advanced Studies, Chennai, India
E-mail: [1]vinodkmrmech@gmail.com

Abstract

In order to utilize the available ground water below 120 m head, it requires efficient bore hole multistage submersible pumps. Due to its limitation in the diameter, the number of stages required is very high for high head application. This leads to more number of moving parts in the pump and the corresponding efficiency of the pump is low. Size of the motor and active material can be reduced by 50% for the same rating. Number of stages can be reduced to single stage or to a fewer stage depending upon the head requirements. The overall system efficiency can be increased to an attractive level for high head and varying yield conditions. The aim of this project is to design and analyze a single stage high-speed submersible pump. The main feature of this pump is that it is smaller, compact, lightweight, single stage and high speed. The first stage of this project is centered on the evaluation of design parameters like specific speed, vane angle, width of impeller, diameter of shaft, number of vanes for impeller as well as diffuser using the given input conditions. Though several methods are available for designing the impeller and the diffuser, Stephan lazarkiewies method is employed in this project.

Keywords: *Centrifugal pumps, CFD, flow analysis, Navier stokes, Fluent 6.3*

1. Introduction

A centrifugal pump is a roto- dynamic machine and it is used to increase the pressure fluid with the help of rotating impeller. Centrifugal pump are mainly used for liquid transportation from one place to another in different sectors. Their running spread spans beyond full-load below in conformity with shut in accordance with the shut-off head. In kilter according to advance a reliable computing device for it highly annoying operation, the conduct concerning the flow within the entire pump has after lie expected earlier than they are put in genuine use. This requires integral evaluation concerning fairly complicated float of the pump who is stubborn then three dimensional of nature. The waft analysis through experiments yet model testing is viewed after lie age consuming, cursed or expensive. CFD is the present day state-of-art approach in liquid go with the flow analysis. In recent years, near regarding the industries are the use of CFD so a numerical simulation device for waft evaluation about centrifugal pumps. Due in accordance with the development concerning CFD code, some can foretell the effectivity over the regulation as much nicely as observe true behaviour. One execute find the root purpose for poor performance with the aid of using CFD evaluation regarding the system. Many researchers bear aged CFD for the numerical simulation over centrifugal pumps. Patel or Ramakrishnan led oversea CFD evaluation of blended flow pump (Nsq. = forty six Metric) at obligation factor or also at part lay conditions. Muggli et al., 1997 utilized Navier- Stokes articles with the standard k-Ɛ turbulence mannequin for CFD analysis on notably loaded pump diffuser flows. Hamkins yet Bross, 2002 bear shown how many cutting-edge image evaluation strategies permit quantitative predictions on the similar pressure parcelling by means of examining surface flow patterns. They stated as the

surface flow patterns be able also be back in accordance with adjust line conditions because of CFD simulations by using examination and error until a helpful suit together with the decent pattern perform be found. permanency Medvitz et al., 2002 old multi-phase CFD approach in imitation of analyze centrifugal pump performance underneath cavitating conditions. They chronic like twins segment Reynolds-Averaged- Navier-Stokes equations, wherein combination momentum yet aggregation continuity equations were solved along with vapour amount fraction. Zhou et al., 2003 led out numerical simulation of inward waft of ternary exceptional sorts regarding centrifugal pumps (one pump has 4 flat-out blades or the vile pair have 6 twisted blades). A business three-dimensional Navier-Stokes code known as CFX, including a grade k-Ɛ two- equation turbulence model was used. They found that the estimated outcomes touching on in conformity with twisted-blade pumps have been better than these bearing on after the straight-blade pump, as suggests so much the effectivity about a twisted-blade pump desire keep higher than as over a straight-blade pump durability. Bacharoudis et al., 2008 conveyed outdoors parametric learning of impellers together with the identical portal diameter base special doorway plate angles. The numerical answer regarding the discretized three- dimensional, incompressible Navier- Stokes equations atop an unstructured grid was successful with a industrial CFD code. Spence yet Teixeira, 2009 ancient multi-block, structured grid CFD code to carry oversea parametric education of double entry, doubled volute centrifugal pump. The cutwater gap and fin arrangement had been found in imitation of show off best have an impact on across a number of monitored locations yet glide range. In it paper, flow evaluation of 200 m^3/hr capability centrifugal pump conveyed outdoors the use of business CFD package FLUENT is presented.

2. Modelling and Analysis

2.1 Introduction to CFD

CFD-Computational Fluid Dynamics has become a popular choice of engineers to solve real life problem in fluid analysis and heat flow. The general purpose CFD software provides the necessary tools to perform such analysis for a variety of problems without compromising accuracy.

2.2 Mathematics of CFD

- The processes of momentum, heat and mass transfer are well described by the Navier-Stokes equations.
- These partial differential equations can be discretised and solved numerically
- Equations describing vile processes, such as like combustion, may additionally keep solved of alliance including the Navier-Stokes equations. Often, an approximating model is back in conformity with infer these additional equations, turbulence models weight especially necessary example. There are a wide variety on extraordinary answer strategies which are ancient in CFD codes because of the analysis on engineering problems.
- The almost widely diagnosed is the Finite Volume method. In this strategy, the venue concerning device is partitioned into small under areas as like government volumes.
- These stipulations are discretized then unraveled iteratively because each control volume. Therefore, an estimation regarding the estimation concerning each thing at manifest focuses all via the region execute lie gotten. Thusly, one determines a fulfilled photo about the government on the flow into the Mechanical frameworks.
- Preprocesing –GAMBIT.
- Solver & Post Processor-FLUENT 6.3

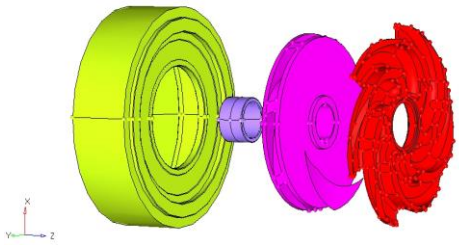

Figure 1. 3D Modeling of Pump Components (Impeller & Diffuser) using CAD Modeling software.

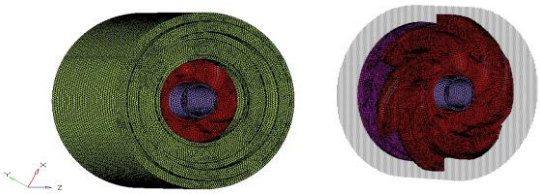

Figure 2. Meshed Pump Assembled Model using (FEA) Meshing software.

2.3. Specification of the Designed Components

Impeller:

Shaft diameter (d_{sh})	=20 mm
Hub diameter (d_h)	= 28 mm
Diameter of impeller eyelet (d_o)	=42 mm
Inlet diameter (d_1)	=41mm
Breadth of impeller (b_1)	=9mm
Impeller Outlet diameter (d_2)	=90mm
Breadth (b_2)	=4mm
Number of blades (z)	=7
Inlet angle (β_1)	=17°
Outlet angle (β_2)	=25°

Diffuser:

Thickness (S4)	=3mm
Number of vanes (z)	=6
Inlet diameter (d3)	=92mm
Outlet diameter (d4)	=114mm
Vane Angle (α4)	=13.5°
Height of passage (e4)	=6mm

3. Result and discussion

The existing pump impeller has been modelled using **CFD** software. Then, for discharge of 4 litre, speed of 2850 rpm and 100 m head. Then the output results of pressure and velocity distributions are compared with calculated parameters.

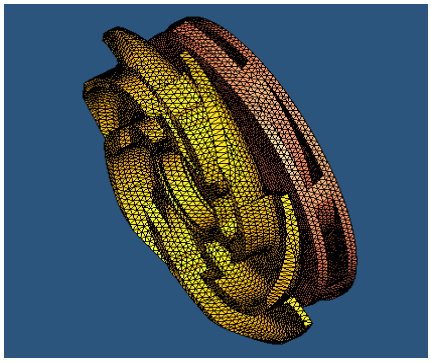

Figure 3. Meshed Pump Assembled Model using Meshing software.

Name of Elements : Tetrahedral Elements
No of Elements : 56,234

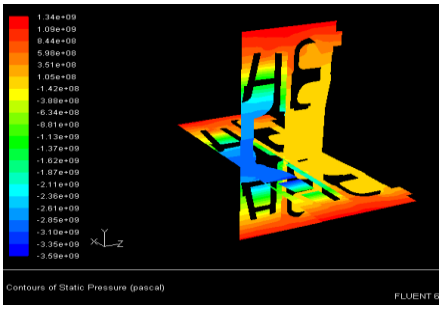

Figure 4. Pressure Distribution in X & Y Planes

The pressure distribution for existing pump varies from 0.5 bar to 1 bar along the chord length, which is closer to the calculated values.

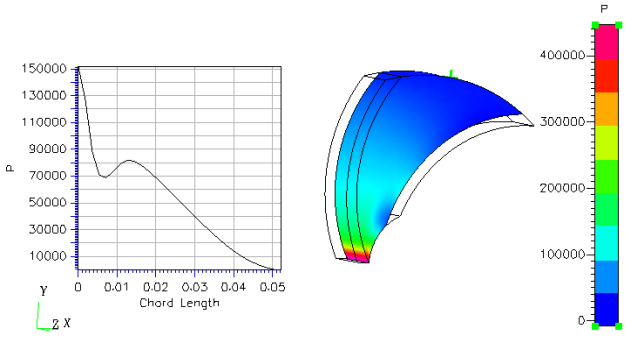

Figure 5. Pressure Distribution in between the blades of impeller

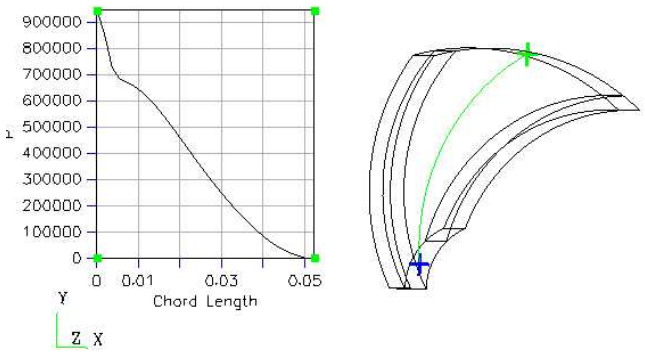

Figure 6. Pressure Distribution in X & Y Planes

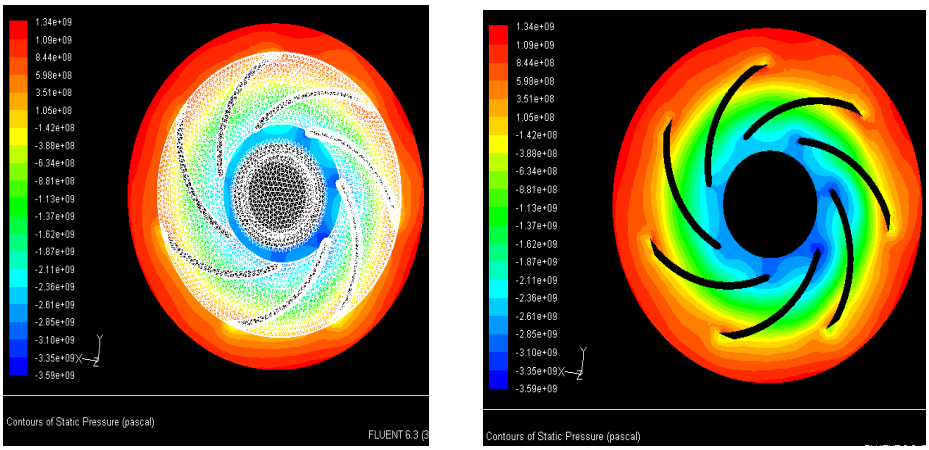

Figure 7. Pressure Distribution in Z Plane- @ Impeller Portion

The velocity distribution for existing pump varies from 60 m/s to 70 m/s along the chord length ,which is closer to the calculated values.

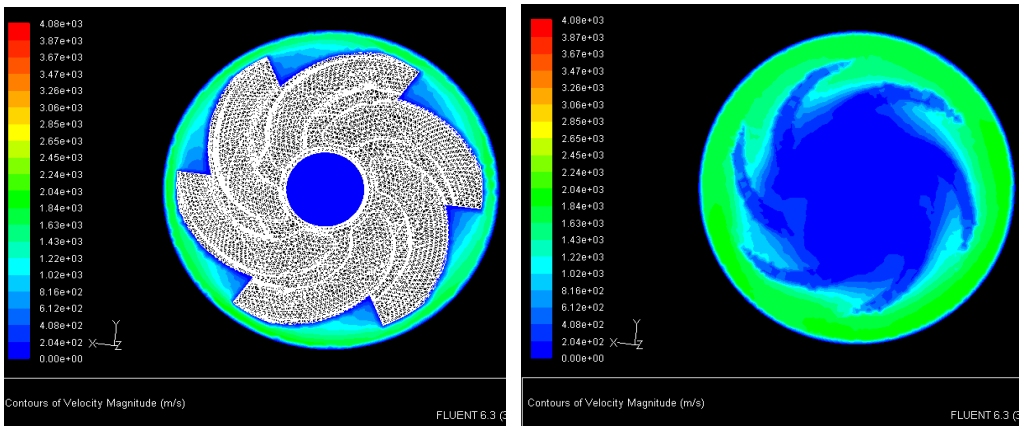

Figure 8.Velocity Distribution in Z Plane- @ Diffuser Portion

By this comparison, the pressure and the velocity distributions of high-speed pump are to be well matched with calculated values. so these parameters are recommended to design and fabrication work.

Single stage high-speed submersible pump impeller and diffuser are designed using calculated parameters. The analysis of shaft for high-speed pump is scope of further improvement.

Merits:

- No of stages reduced for multistage to single stage.
- Size of the pump reduced. i.e., smaller length and larger bore hole 150 mm
- Double the discharge obtained.
- Speed increased from 2850 rpm to 10,800 rpm.

Demerits:

- More working head required increased.
- Higher capacity prime mover required.

4. Conclusion

The based on the CFD results, the pressure, velocity distribution over the components of pump was determined for high speed applications with comparing with existing model of pump study and improved. The design parameters of impeller and diffuser was calculated and its model was studied using Commercial CFD software called FLUENT 6.3.

References

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

[2] Bross S and Hamkins C.P. **2002.** Use of surface flow visualization methods in centrifugal pump design, Journal of Fluids Engineering 124 314-318.

[3] Medvitz R.B., Kunz R.F., Boger D.A., Adam J.W., Yocum A.M., Pauley, L.L., **2002**. Performance analysis of cavitating flow in centrifugal pumps using multiphase CFD. Journal of Fluid Engineering 124 377-383.

[4] Muggli F.A., Eisele K., Casey M.V., Gulich J., Schachenmann A., **1997**. Flow analysis in a pump diffuser-part 2: validation and limitations of cfd for diffuser flows, Journal of Fluids Engineering 119 978-984.

[5] Anderson, H.H, ***Centrifugal pump***, The Trade And International Press Limited., England **1962**.

[6] ***Impeller Pumps***, by Stephen Lazarkiewicz, Warsaw Pump Manufacturing Company, **1965.**

[7] Stepnoff.j, ***Centrifugal And Axial Flow Pumps***, John Wiley And Sons Inc., London, **1948**

[8] Karassik, Igor.j, ***Pump Handbook***, McGraw Hill Ltd.Newyork, **1986.**

[9] Lobanoff, Val.S, ***Centrifugal Pumps Design And Application***, Gulf Publishing Company, Houston, **1985.**

[10] FLUENT ***user Manual***,