Design And Analysis of Centrifugal Pump Impeller Using Computational Fluid Dynamics

M.Maniyarasan[1], S.Prasanth[2], T.Premchand[3], Sheikh Mohamed Ovysi[4], R.Ragavan[5]
[1]Professor, Dept of Mechanical Engineering
[2, 3, 4, 5]Dept of Mechanical Engineering
[1, 2, 3, 4, 5] Muthayammal Engineering College, Namakkal

Abstract- *Centrifugal pumps are used extensively for pumping water over short to medium distance through pipeline where the requirements of head and discharge are moderate. This project is devoted to enhance the performance of the centrifugal pump through design modification of impeller. Theories on pump characteristics are studied in detail. Vane profile of the impeller is generated using point by point method. The impeller is modelled in Solidworks 2012 software and CFD analysis is done using fluid flow simulation package. CFD analysis enables to predict the performance of the pump and a comparative analysis is made for the entire control volume by varying meshing.*

Keywords- centrifugal pump, impeller analysis, pump performance, pump characteristics, CFD analysis

I. INTRODUCTION

A pump is a mechanical device for moving a fluid from a lower to a higher location, or from a lower to a higher pressure area. Mechanical energy is given to the pump and it is then converted into hydraulic energy of fluid. Pumps produce negative pressure at the pressure at the inlet so that the atmospheric pressure pushes the fluid towards the pump. The fluid coming into the pump is pushed the towards the outlet mechanically where positive pressure is generated. Pumps are classified in number of the ways according to their purpose, specifications, design, environment etc.

II. TYPES OF CENTRIFUGAL PUMP

Radial Flow Pumps

These pumps are often simply referred to as centrifugal pumps. The fluid enters along the axial plane, is accelerated by the impeller and exists at right angles to the shaft (radially). Radial flow pumps operate at higher pressures and lower flow rates than axial and mixed flow pumps. The radial flow pumps, by its principle, are converse of the Francis turbine. The flow is radially outward, and the hence the fluid gains in centrifugal head while flowing through it. Because of certain inherent advantages, such as compactness, smooth and uniform flow, low initial cost and high efficiency even at low heads, centrifugal pumps are pumps are used in almost all pumping systems.

Axial flow pumps

Axial flow pumps differ from radial flow in that the fluid enters and exits along the same direction parallel to the rotational shaft. The fluid is not accelerated but instead ―lifted‖ by the action of the impeller. They may be linked to a propeller spinning in the length of the tube. Axial flow pumps operate at much low pressures and higher flow rates that radial flow pumps.

Mixed flow pumps

Mixed flow pumps, as the name suggests, function as a compromise between radial and axial flow pumps , the fluid experiences both radial acceleration and lift and exists the impeller somewhere between 0-90 degrees from the axial direction. As a consequence mixed flow pumps operate at high pressure than axial flow pumps while delivering higher discharges than radial flow pumps. The exit angle of the flow dictates the pressure head discharge characteristic in relation to radial and mixed flow.

General characteristics of centrifugal pump

- Every centrifugal pump has two characteristics that are same; each has an impeller that forces the liquid being pumped into a rotary motion, each has a casing, which directs the liquid to the impeller. The liquid leaves the impeller as the impeller rotates.
- The liquid leaves with high velocity and pressure than it had when it entered. There is a conversion of some of the velocity to pressure than takes place before the liquid leaves the pump; this partial conversion takes place in the pump casing. Head loss is associated with conversion and must be taken into account.
- The size of the impeller and the pump casing vary greatly with the type of centrifugal pump. Centrifugal

pumps are often classified by a type number known as the specific speed that varies with the shape of the impeller. Two main components of a centrifugal pump are the impeller and the casing. The impeller is a rotating component and the casing is a stationary component .The impeller is a rotating component. In centrifugal pump, water enters axially through the impeller eyes and water exits radially. The pump .Casing is to guide the liquid to the impeller, converts into pressure the high velocity kinetic energy of the flow from the impeller discharge and leads liquid away of the energy having imparted to the liquid comes from the volute casing.

- In a centrifugal pump, the liquid is forced by atmospheric or other pressure into a set of rotating vanes enclosed within a housing or casing that is used to impart energy to a fluid through centrifugal force. The design and performance analysis of radial flow centrifugal pump are chosen because it is the most useful mechanical roto dynamic machine in fluid works which is widely used in domestic, irrigation, industry, large plants and river water pumping system.

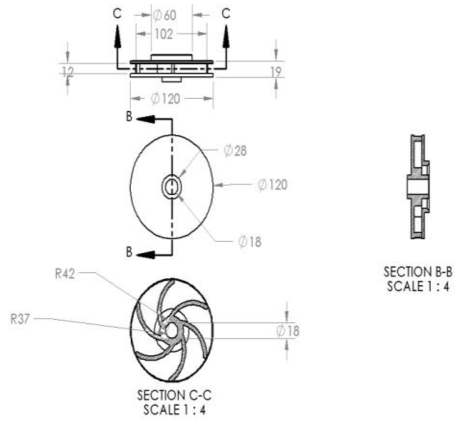
III. OBJECTIVE

- To design a centrifugal pump impeller for the given specification.
- To provide design methodology for centrifugal pump impellers.
- Proposed for the Partial modification of the blade flow passage which affects the entire flow field.
- To provide centrifugal impellers that can operate more efficiently and quietly.
- To use a commercial CFD software to find the change in performance from initial design.

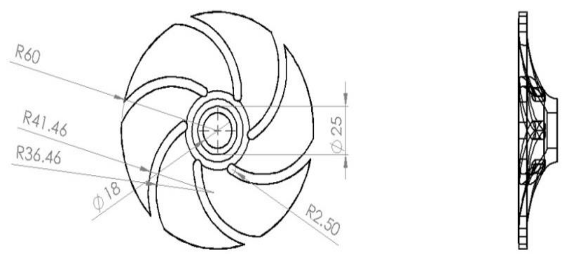
Design Specification

Inlet Dia (D_1)	50 mm
Outlet Dia (D_2)	120 mm
No. of Vanes (z)	6 nos.
Vane Outlet Angle (φ)	32^0
Vane Radial Length (l)	40mm
Vane inlet width (b_1)	17.5 mm
Vane outlet width (b_2)	8.5 mm
Vane thickness (e)	5 mm

Detailed view of the pump

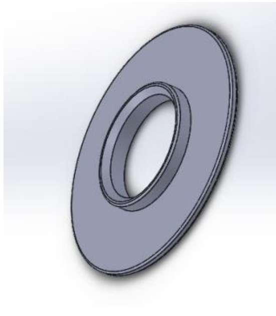


Impeller Dimensions



3D model of impeller

Three-dimensional model of an impeller was first created in Solidworks 2012 software and exported into STEP files. The STEP files were then imported into fluid flow simulation, the mesh generator. The fluid volume was split into a rotating fluid volume, a scroll volume , an inlet cone volume, and an inlet/outlet duct volume. The inlet and outlet ducts were intentionally set to simulate the actual measuring situation and to provide better boundary conditions for simulations.



Impeller Plate

IV. INTRODUCTION OF CFD

Computational fluid dynamics (CFD) uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow (the Navier- strokes and allied equations), for predefined geometries and boundary conditions. The result is a wealth of the predictions for flow velocity, temperature, density, and chemical concentrations for any region where flow occurs.

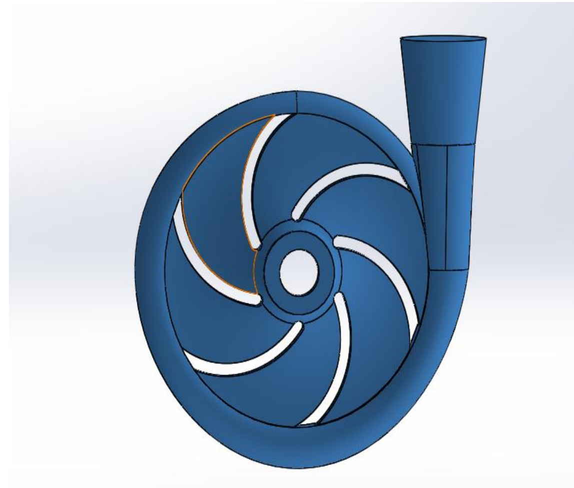
CFD analysis begins with a mathematical model of a physical problem, conservation of matter, momentum, and energy must be satisfied throughout the region of interest. Fluid properties and modeled empirically. Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, in viscid, two dimensional).

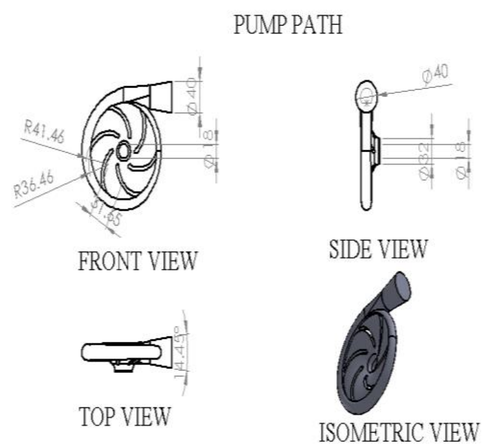
Provide appropriate initial and boundary conditions for the problem. CFD applies numerical methods (called discretization) to develop approximations of the governing equations of fluid mechanical in the fluid region of interest. The solution is post processed to extract quantities of interest (e.g. lift, drag, torque, heat transfer, separation, pressure loss, etc.).

Practical advantages of employing CFD

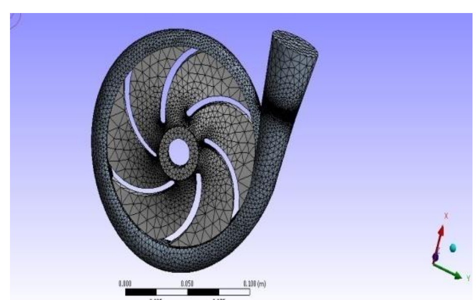
- The followings are among the many reasons why CFD is being widely used today
- CFD predicts performance before modifying or installing the systems
- Without modifying and/or installing actual systems or prototype, CFD can predict what design change is most crucial to enhance performance
- CFD provides exact and detailed information about HVAC design parameters

CFDflow path 3d assemble diagram





Meshing



Meshing is defined as the process of dividing the whole component into a number of elements so that whenever the load is applied on the component it distributes the load uniformly called as **meshing**. A component is analyzed in two ways. One is with **Meshing** and the other is without **meshing**.

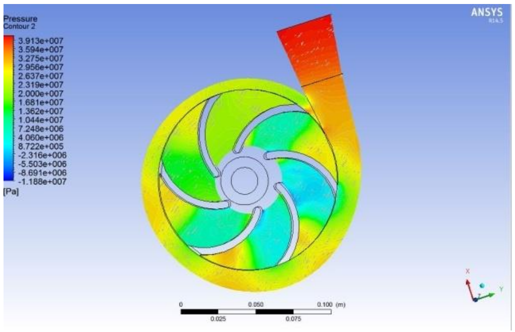
Mesh generation is the practice of creating a **mesh**, a subdivision of a continuous geometric space into discrete geometric and topological cells. ... Meshes are used for rendering to a computer screen and for physical simulation such as finite element analysis or **computational fluid dynamics**.

SimScale offers four primary meshing methods:

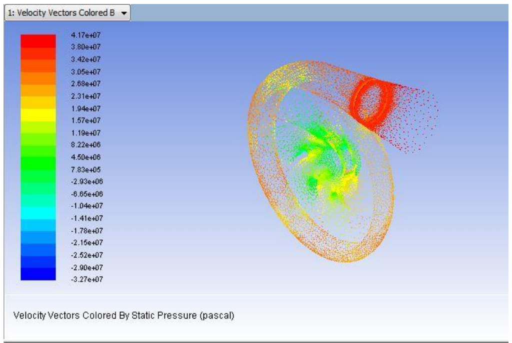
- Tet-dominant
- Hex-dominant automatic (only CFD)
- Hex-dominant parametric (only CFD)
- Hex-dominant automatic “wind-tunnel/external flow” (only CFD)

V. RESULTS

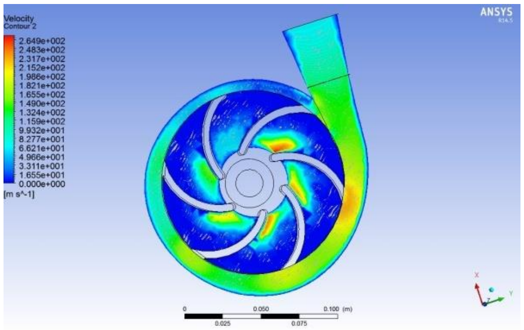
1. PRESUURE



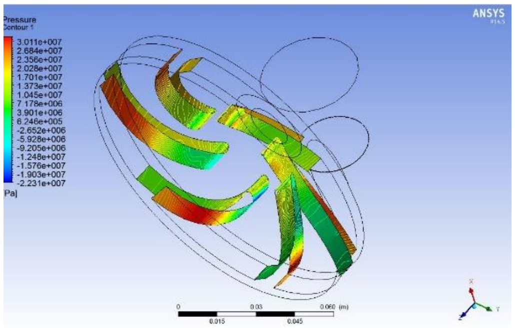
2. VECTOR PRESSURE



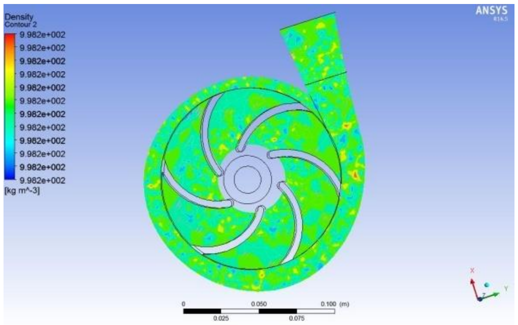
3. VELOCITY



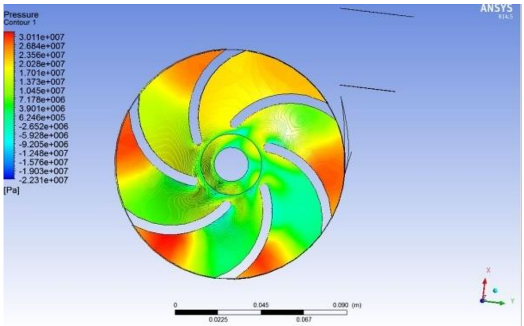
4. Blade Pressur



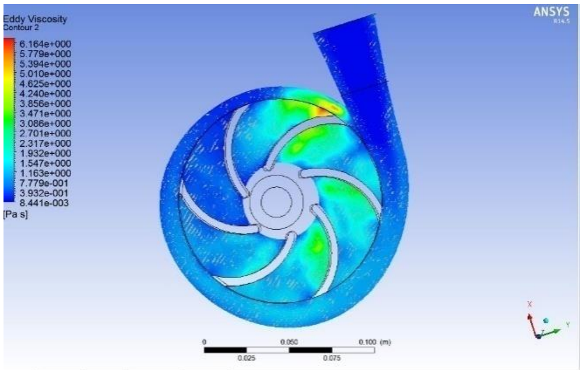
5. DENSITY



6. IMPELLER PRESSURE



7. EDDY VISCOCITY



VI. CONCLUSION

Pump impeller is designed for the given specification and numerical analysis is carried out in fluid flow simulation package. Contour plots are also obtained for the distribution of static pressure, velocity and wall shear stress. The following are the performance evaluation of the pump, arrived from CFD study. Overall efficiency of the pump is 61%, CFD results predict total head of 50 m.

REFERENCES

[1] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris (2008), "Parametric investigation by way of the radial siphon outlet differ sharply", building mechanical open diary, 2008, 75-83.

[2] LIU Houlin, WANG Yong, YUAN Shouqi, TAN Minggao, and WANG Kai (2010), "The Influence of the amount of the reducing part regular to suck radial", diary China mechanical engineering in 2010

[3] Three. B.Mohan, B.E. Kumar (2014), "Analysis of Auxiliary at Impeller of Axial Flow Compressor makes use of Fem", International Journal of Engineering Research and Technology ISSN: 2278-0181 Vol. Three Issue 10.

[4] Four. S.Rajendran and Dr.K.Purushothaman, (2012), "Investigation of outer impeller siphon utilising ANSYS CFX-", International Journal of Research and Technology, Vol 1 Issue three.

[5] Syam Prasad, BVVV LakshmipathiRao, A Babji, Dr.P Kumar Babu, (2013), "Static and Dynamic Analysis of Centrifugal Pump Impeller", International Journal of Scientific and Engineering Research, Volume 4, Issue 10.

[6] Neelambika, Veerbhadrappa (2014) "CFD investigation of mixed float impeller", diaries International layout research and innovation.

[7] SambhrantSrivastavaa, Apurba Kumar Roy and Kaushik Kumar, (2014) "Plan siphon combined glide impeller and approval exploit FEM examination", Science Direct, Procedia Technology 14 PN 181-187.

[8] SantoshShukla, Apurba Kumar Roy and Kaushik Kumar, (2015) "Material Selection for the pointy fringe of a Mixed Flow Pump Impeller Using ANSYS", Science Direct, Materials Today: Proceedings 2, 2022-2029.

[9] Basawaraj, H.Hasu (2016), "Examination investigation into the siphon gushing manage vane impeller with FEM method by way of making use of the everyday ingredients", IJES 2016 Volume 2016 Issue 8

[10] S.Rajendran and Dr.K.Purushothaman, "Examination of the impeller diffusion siphon utilizing ANSYS CFX-," International Journal of Engineering Research and Technology, Vol. 1, Issue 3, 2012.

[11] S R Shah, S V Jain and V J Lakhera, "CFD go with the flow research based totally siphon diffusion," Proceedings of the 37th National and fourth International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, 2010.

[12] P.UshaShriansC.Syamsundar, "computational examination at the execution of the impeller diffusion siphon," Proceedings of the 37th National and fourth International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, 2010