E3S Web of Conferences **391**, 01088 (2023) https://doi.org/10.1051/e3sconf/202339101088
ICMED-ICMPC 2023

Computational fluid dynamics (CFD) analysis of centrifugal pumps

Ruchir nandan [1],*Syed* Mohammed Pasha [1],*D.* Ashish [1], and *B.*Ch. Nookaraju [1*]

[1]Department of Mechanical Engineering, Gokaraju Rangaraju Institute of Engineering and Technology, Hyderabad, India – 500090.

Abstract: Centrifugal pumps are mostly used in different fields like industries, agriculture and domestic applications. Objective of this paper is to give a critical review of CFD analysis of centrifugal pump along with future scope for further improvement of flow efficiency. Computational Fluid Dynamics is the most used tool for simulation and analysis. 3-D numerical CFD tool is used for simulation of the flow field characteristics inside the pump machinery. CFD for centrifugal pump is used to solve numerical simulation problems working as a tool for getting performance prediction of modelled design at different conditions, and can derive information and study of pump performance on the system, Study of pressure contours, velocity contours, flow streamlines, cavitation analysis, analysis of interaction effects in different components, prediction of axial thrust etc., is also be studied by CFD techniques. Simulation makes it possible to visualize the flow condition inside a centrifugal pump. The present paper describes the head, power, efficiency and to evaluate the pump performance using the ANSYS CFX-14, a computational fluid dynamics simulation tool.

Keywords: Centrifugal Pump, CFD, Improved efficiency

1 Introduction

The main aim of the project is to improve the efficiency of centrifugal pump by makes some of the changes in the standard like changing the number of Blades and blade angle. A Centrifugal pump is a rotary machine that transforms kinetic energy into the pressure head of the fluid. External power from an electric motor or diesel generator turns the pump impeller. Then, under the influence of centrifugal force, the fluid entering the impeller reaches its tip and leaves the volute casing. The most popular type of pump utilized in businesses, agricultural, and household settings is the centrifugal pump. The rotating impeller of the centrifugal pump transfers mechanical energy from the motor to the fluid, increasing the pressure. The liquid enters through the impeller's eye and travels along the blades. Due to the impeller's rotation, centrifugal force develops in this situation. Due to the increased

[*] Corresponding autor: bulipechinanookaraju@gmail.com

fluid velocity caused by the centrifugal force, kinetic energy is converted to pressure. The various centrifugal pump components are depicted.

2 Literature review

2.1 papers reviewed for validation

2.1.1 Khin Cho Thin et al's paper

Khin Cho Thin et al, analysed the performance parameters of a 1 HP centrifugal pump rotor with nine blades. This document covers the turbine design process in great detail. For the purpose of developing the impeller, they used the BERMAN method. There is a description of the various elements that influence the pump's performance. Designing the blade begins with determining the pump's particular speed. Specific speed, a conventional dimensionless number, is the rate at which one gallon of water is transported at a head of one foot. The volume, water, and shaft power calculations were then described. The efficiency of the motor, compressor, and hydraulic system all affect the engine's efficiency. The methods for determining the hub length, shaft diameter, inlet and outlet flow angles, impeller, hub diameter, and with the aid of the nine-blade impeller, outlet sites were explained. Vanes' intake and exit flow angles as well as the inlet velocity triangles were displayed in the review. The efficiency of the pump was determined by plotting the pump's head output over a variety of flow rates. [1]

2.1.2 Voorde etal's paper

Voorde etal, used the CFD program (FLUENT) to predict how their centrifugal pump would work. Two pumps were evaluated, one of which had a low specific speed, and yet another with medium-speed precision. The fact that FLUENT provides there was noted. The Multiple Reference Frame method is one of the flow-solving strategies used in turbomachinery. Sliding Mesh (SM), Mixing Plane (MP), and MRF methods, utilizing all three methods. The movement in the rotor was calculated using an absolute reference frame, while the stator was determined using a rotating reference frame. In contrast to the first two techniques, which solved steady flow equations, the sliding mesh method solved unsteady flow equations. [2]

2.1.3 S.H.Winoto et al's paper

S.H. Winoto et al looked into a centrifugal pump's irregular flow. It was determined to use a centrifugal pump with a six-blade impeller that was twisted. The study made use of the turbulence model with k- 2 equations. When modelling, various frameworks of reference were used. Tetrahedral mesh contact was achieved by joining the impeller and the volute housing using the sliding mesh technique. The study revealed a recirculation zone near the suction end. The flow that was discharged from the propeller into the volute was observed to have a spiral flow pattern. [3]

2.1.4 S. Rajendran's paper

S. Rajendran performed similar experiments in the analysis of a centrifugal pump impeller using ANSYS-CFX, which was used to construct a three dimensional, completely turbulent model of the compressible flow across the geometry of an impeller. A six-blade impeller with a similar design was used in the research. The geometry and mesh of the impeller region were produced using Ansys Workbench. Tetrahedral cell-containing sections of the unstructured grid were used to build the impeller and volute. Hexahedral cells with an organized framework were created around the blades to obtain more boundary layer characteristics. When determining the boundary layers, the impeller was viewed as a rotating frame of reference with a rotational speed. Rotating fluid was hot water that was 25 °C. [4]

Through the validation of the above papers the centrifugal pump of 7 blades have given higher efficiency when compared with centrifugal pumps of 5,9 and 11 blades. The efficiency of the centrifugal pump can also be optimized by changing the blade angles and dimensions of the centrifugal pump. The efficiency of the centrifugal pump also differs on factors like material used, cavitation, mass flow rate, and speed of the impeller. In this analysis the efficiency of the centrifugal pump is enhanced by changing the blade angles and the geometry of the impeller blade by standardising the mass flow rate, rise in head and the power supplied as the boundary conditions.

3 Numerical formulation & Equations

All statistical studies were carried out through simulation using ANSYS tools. The pump efficiency (η) with NPSH was established as a major objective function for instantaneous optimization in the designated pumps. Efficiency of the ideal centrifugal pump was described as below

$$\eta = \frac{P_{in}}{P_{out}} \quad (1)$$

$$P_{out} = \rho \times g \times H \times Q \quad (2)$$

$$\text{NPSH} = \frac{P_{in} - P_{out}}{Y} - \frac{v_{in}^2}{2g} \quad (3)$$

Where P_{in} stands for the inlet pressure, Y and v_{in} stand for the specifically unsolidified mass and charge velocity, respectively, and Q is charge flow rate where P_{in} stands for the inlet pressure and v_{in} stand for the specifically unsolidified mass and charge velocity, respectively, and Q is charge flow rate .

4 Materials

Cast iron is the material that is used in the centrifugal pump illustration. Cast iron makes up the hub, the volute, and the impeller blades and water are the fluid utilized to simulate the centrifugal pump, as we are still trying to research on creating different model using different base materials example like aluminium, and different alloys in ANSYS.

5 Geometry

The single stage centrifugal pump is designed and modelled in ANSYS designer modeller. The impeller blades and the volute are designed separately. The volute, impeller blade, impeller and the trailing edge of the impeller blade is shown in fig1.1, fig1.2, fig1.3 and fig1.4 respectfully.

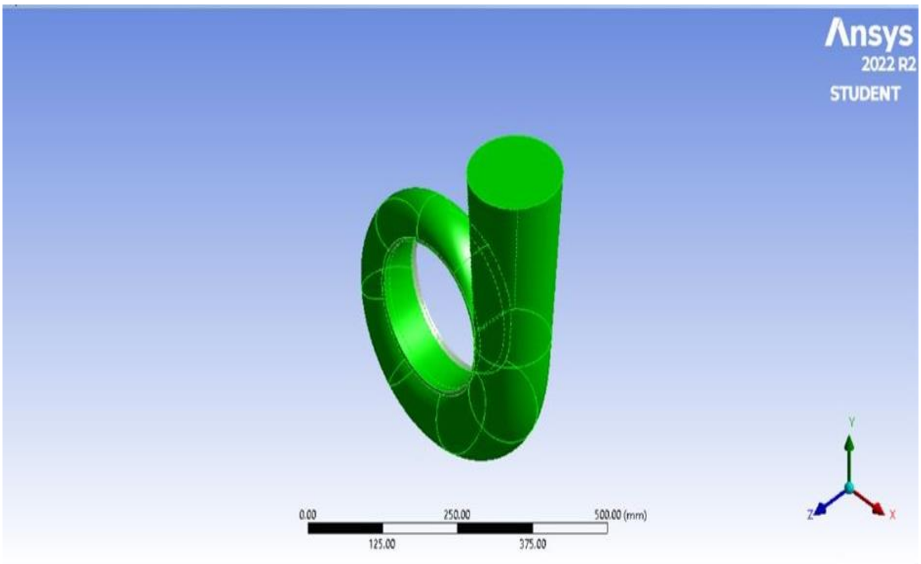

Fig.1 Volute

The volute of a centrifugal pump is the casing that receives the fluid being pumped by the impeller, maintaining the velocity of the fluid.

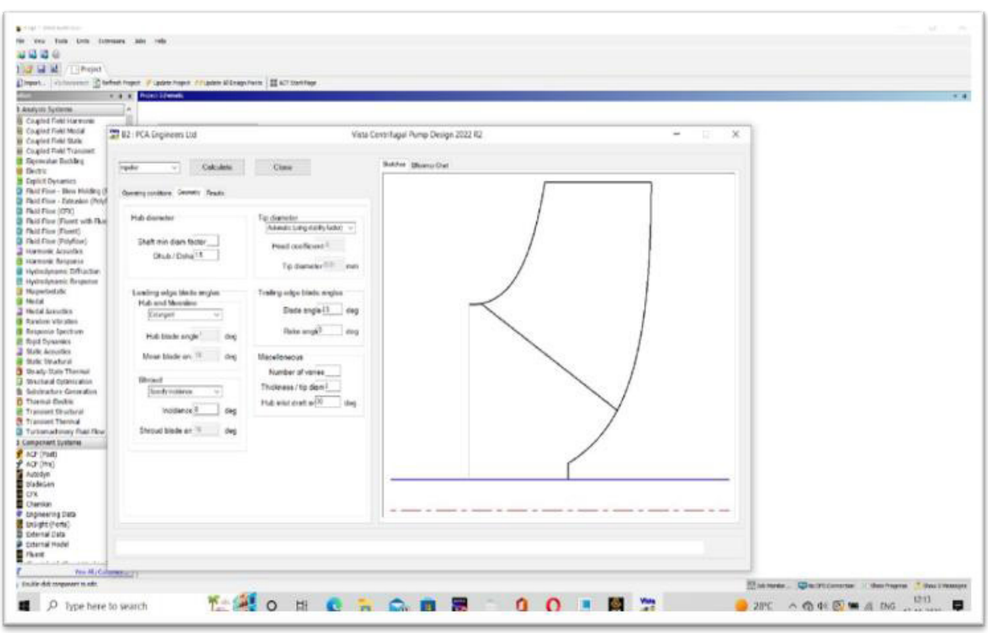

Fig. 2 Impeller blade

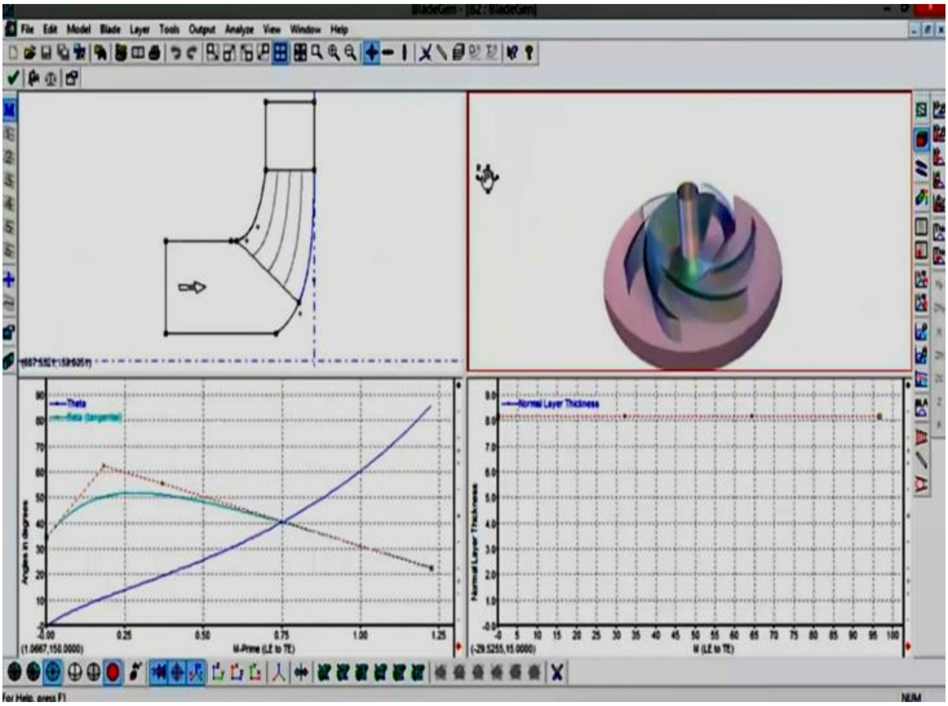

Fig 3 Impeller

the rotating part of a centrifugal pump, designed to move a fluid by rotation.

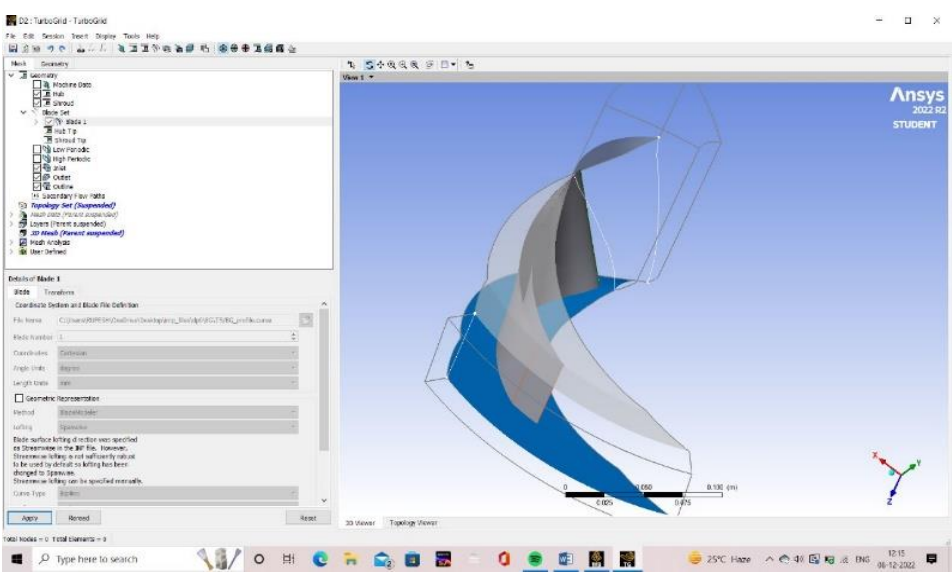

Fig. 4 Impeller blade

6 Meshing

The technique of meshing in ANSYS involves cutting down the entire body into tiny linear cubes or tetrahedral structures. The Navier-Strokes equation may be used to easily solve these smaller cubes or tetrahedral shapes. The impeller and volute geometry of this centrifugal pump are divided into tetrahedral shapes as illustrated in the figure.

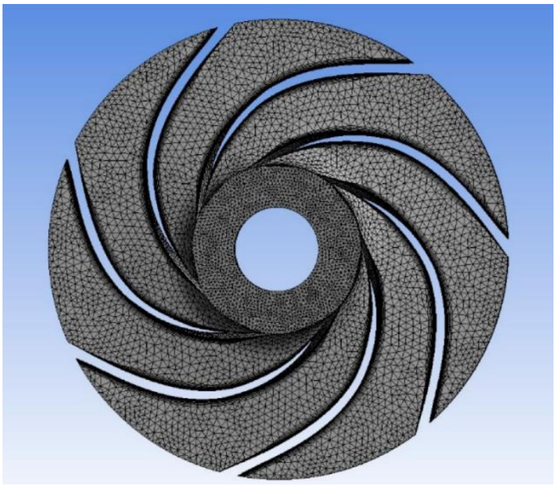

Fig.5 Meshing of the impeller of the centrifugal pump

Fig.6 Meshing of the volute of the centrifugal pump

7 Boundary conditions

The volume flow rate, the fluid density, the head rise, the intake flow angle, and the power applied are the parameters that make up the centrifugal pump's boundary conditions. The volume flow rate is configured at 280m^3/ hr to correspond to commercial use in industries and agricultural areas. The water's 1000 kg/m^3 density serves as the fluid's density. The centrifugal pump's target head increase is set at 20m. The configuration of the intake flow angle is 90 ^{o} (degrees). The pump is powered by a 19-kW electricity supply.

8 Result and discussion

The blades have been designed with different angles and trailing edges angles the figures below shows the designed blade with their maximum efficiency at a suitable speed (in RPM)

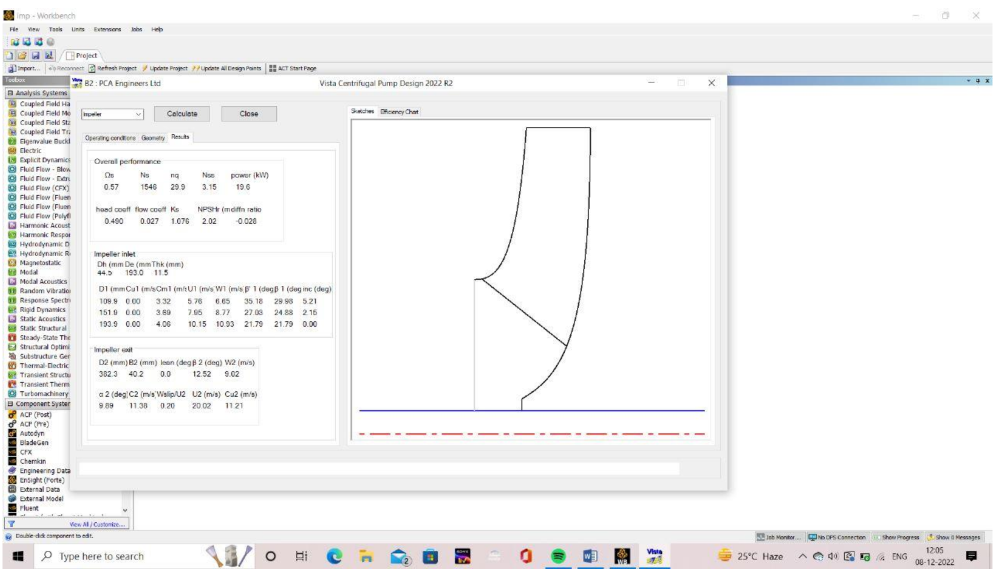

Fig.7 Blade angle of 20.5 degrees

When the impeller rotates at a speed of 1000 rpm with an input power of 19 kW, the impeller blade achieves its maximum efficiency of 79% for head rise of 20m. The blade angle of the above design is 20.5 degrees with trailing edge angle of 2.5 degrees

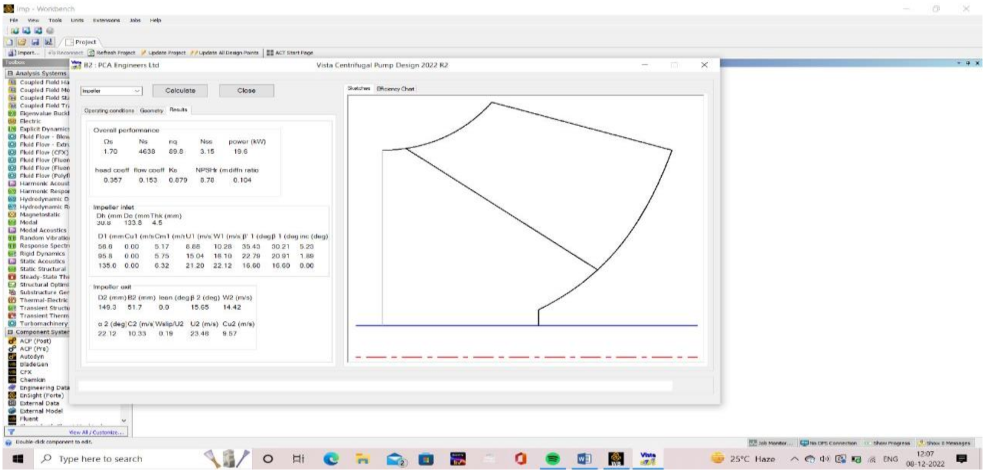

Fig 8 Blade angle of 26 degrees

When the impeller rotates at a speed of 2100 rpm with an input power of 19 kW, the impeller blade achieves its maximum efficiency of 82% for head rise of 20m. The blade angle of the above design is 26 degrees with trailing edge angle of 2.5 degrees

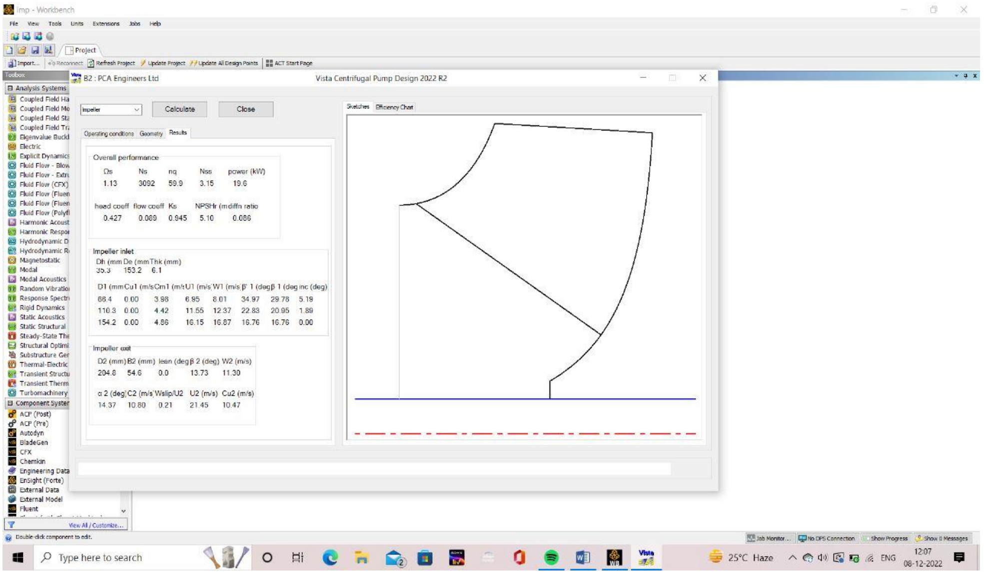

Fig.9 Blade angle of 25 degrees

When the impeller rotates at a speed of 2000 rpm with an input power of 19 kW, the impeller blade achieves its maximum efficiency of 84% for head rise of 20m as shown in the figure10. The blade angle of the above design is 25 degrees with trailing edge angle of 2.5 degrees

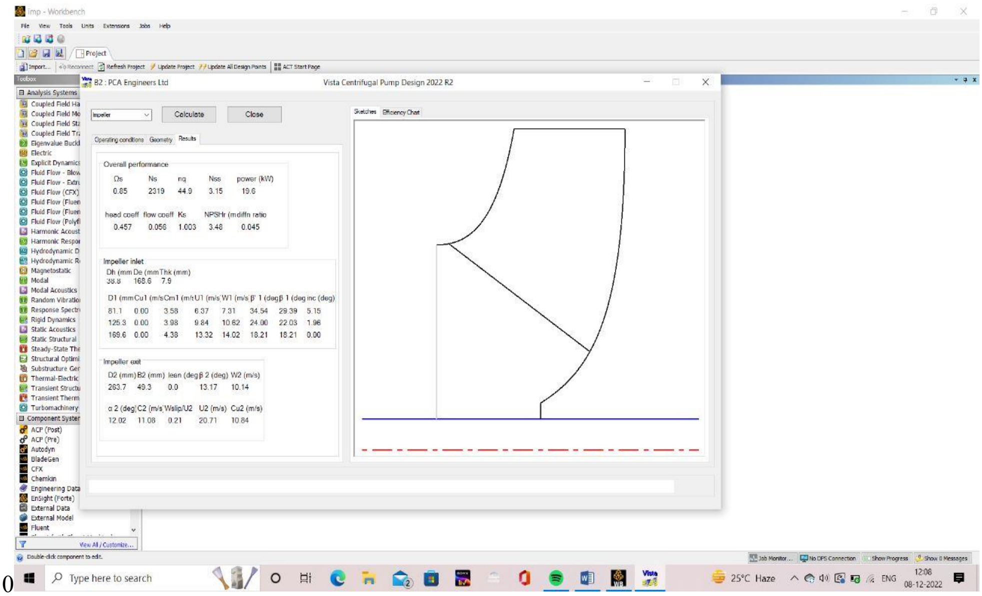

Fig.10. Blade angle of 23 degrees

When the impeller rotates at a speed of 1450 rpm with an input power of 19 kW, the impeller blade achieves its maximum efficiency of 91% for head rise of 20m as shown in the fig.10. The blade angle of the above design is 23 degrees with trailing edge angle of 2.5 degrees

8.1 Efficiency chart

Fig.11 Efficiency chart

efficiency chart of the modelled centrifugal pump with efficiency on the Y-axis and head coefficient on X-axis as shown in the fig.11

a. Hydraulic efficiency = 0.874 = 87.4%
b. Volumetric efficiency = 0.97 = 97%

 https://doi.org/10.1051/e3sconf/202339101088

c. Overall pump efficiency = 0.804 = 91.4%

9 Contours

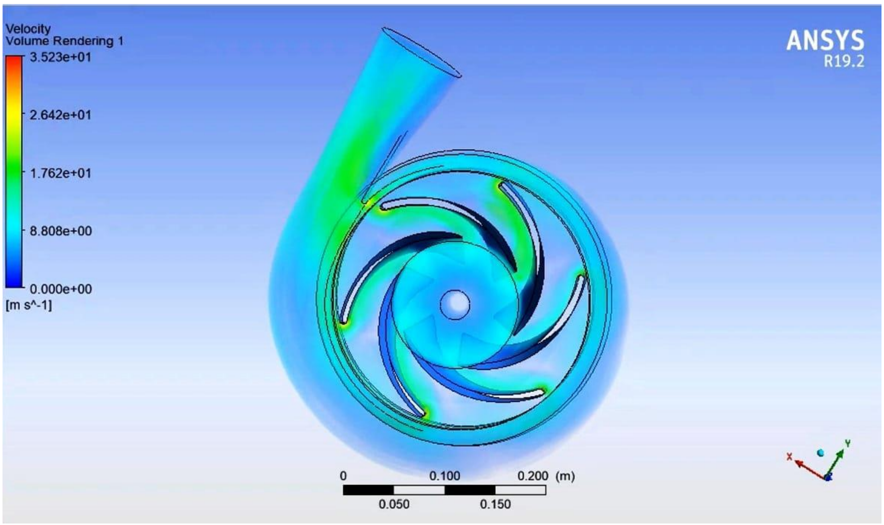

Fig 12 Velocity contour.

velocity contour of the modelled centrifugal pump with maximum velocity of 35.23m/s and minimum velocity of 8.808m/s as shown in the fig.12

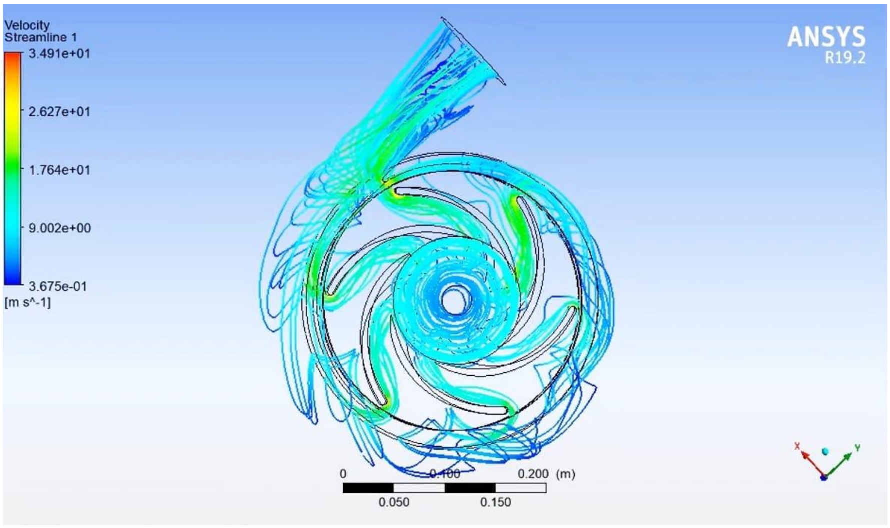

Fig.13 Streamline.

Streamline of the water flowing through the centrifugal pump with maximum velocity of 34.91m/s and minimum velocity of 0.3675m/s as shown in the fig.13.

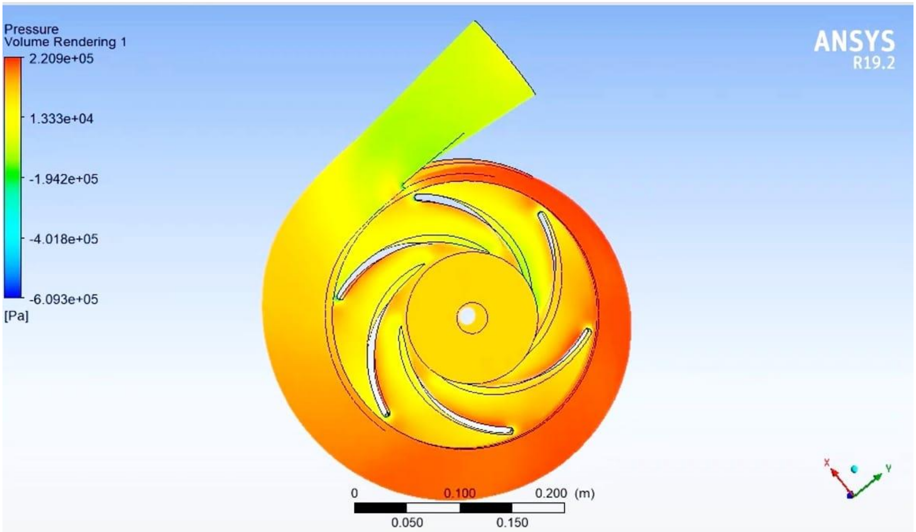

Fig 14 Pressure contour

Pressure contour of the modelled centrifugal pump with maximum pressure of 2.209×10^5 Pa and minimum pressure of 1.942×10^5 Pa as shown in the fig.14.

9 Efficiency Factor

Noise, vibration, recirculation, and heat are some of the factors that impede the pump from operating at a higher efficiency. Other factors that reduce pump effectiveness include:

9.1 Mechanical Efficiency

For the different kinds, such as centrifugal vacuum pumps and others, the formula for this efficiency is the ratio of the actual power given to the pump to the theoretical power the pump needs to operate. In this situation, we have used the efficiency formula to determine the amount of power wasted in particular moving components, like the bearings. In the end, the mechanical efficiency values are noted and done some changes to improve the efficiency.

9.2 Hydraulic Efficiency

centrifugal pumps and other types of pumps transform mechanical energy into hydraulic energy. Flow, pressure, and velocity make up this. The mechanical energy given to the rotor divided by the useable hydrodynamic energy in the form of fluid is the ratio formula for hydraulic efficiency. The efficiency of our modelled centrifugal pump has been raised typically when we had made some change in blade angle, and the results has been noted down.

9.3 Volumetric efficiency

The output of centrifugal pumps' actual power to their input of actual power is the ratio formula for calculating overall efficiency. We would base our calculation of the total amount of energy lost on the overall efficiency.

10 Comparison Table

This table 1 contain all the information, which is used to compare the results of our self-design model centrifugal pump with a real commercial pump present in the market

Table 1 comparison between commercial pump and modelled pump

Contents	**Commercial pump**	**Modelled pump**
RPM	1500 rpm	1450 rpm
Mass Flow Rate	200 m^3/hr	280 m^3 /hr
Density Of the Fluid	1000 kg/m^3	1000 kg/m^3
Inlet Flow Angle	90 degrees	90 degrees
Blade Angle	25	23
No Of Blades	7	7
Inlet Diameter	51 mm	51 mm
Outlet Diameter	117 mm	117 mm
Head Range	12.8 m	17.6 m
Trailing Edge Blade Angle	3.2 degrees	2.5 degrees
Hub Diameter	190 mm	190 mm
Impeller Diameter	286 mm	286 mm
Outlet Width	37 mm	37 mm
Blade Outlet Angle	23 degrees	22 degrees

 https://doi.org/10.1051/e3sconf/202339101088

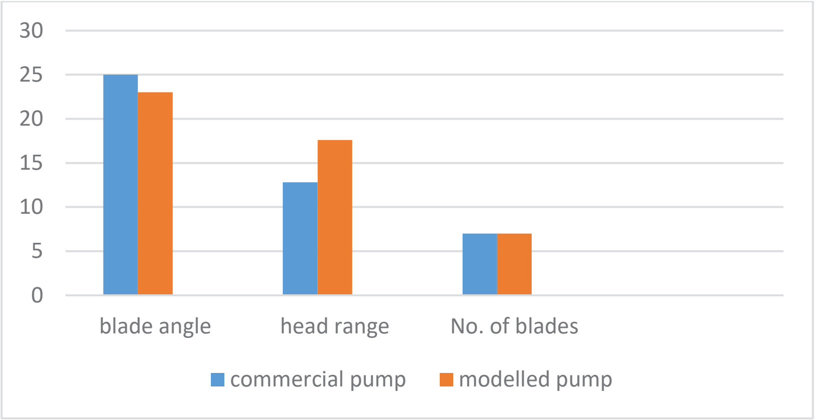

Fig 15 comparison of commercial and modelled centrifugal pump.

The blade angle head rise, and number of blades have been compared. In Blade angle point of view commercial pump was more compared with modelled pump as shown in the fig.16. During head range model pump was more efficient than commercial pump as shown in the fig.15

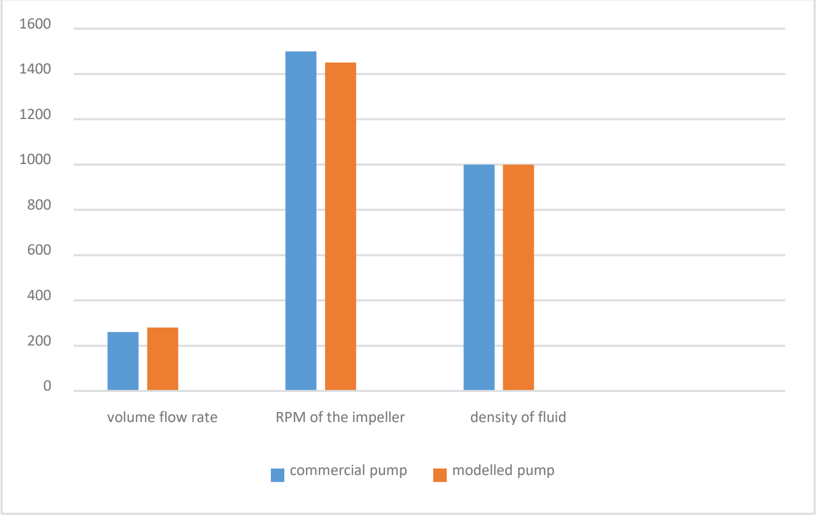

Fig 16 Comparison of commercial and modelled centrifugal pump.

The volume flow rate, the RPM of the impeller and the density of the fluid was compared as shown in the fig.16. As the RPM increases flow rate increases as show in fig.16.

11 Conclusion

The centrifugal pump is designed in ANSYS where the impeller and the volute are designed separately. The impeller blades have been modified to produce different efficiencies. The blade design with the boundary conditions like volume flow rate, density of the fluid, the power supplied, and the target head rise which are similar to the industrial necessities, the blade with blade angle of 23 degrees and the trailing edge angle of 2.5 degrees tended to reach the target head rise. The head produced by the modelled centrifugal pump is 17.6m for a volume flow rate of 280m^3/hr, for electricity supply of 19k-W, and the density of the fluid(water) is 1000kg/m^3. The velocity and pressure contours have been generated along with the streamline. The efficiency obtained by the modelled centrifugal pump is 91%

References

1. Thin, Khin & Khaing, Mya & Aye, Khin. "Design and Performance Analysis of Centrifugal Pump" Vol.46 (2008).
2. E. Dick, J. Viereneels, S. Serbruyns,, and J. Voorde, "Performance Prediction of Centrifugal Pumps with CFD-Tools," Task Quarterly, vol. 5, pp. 579-594 (2001).
3. Cheah, K. & Lee, T. & Winoto, S. & Zhao, "Numerical Flow Simulation in a Centrifugal Pump at Design and Off-Design Conditions". International Journal of Rotating Machinery 10.1155/2007/83641(2007).
4. S.Rajendran, Dr.K.Purushothaman, "Analysis of a centrifugal pump impeller using ANSYS-CFX" Vol. 01 Issue (03 May 2012).
5. Weidong Zhou, Zhimei Zhao, T. S. Lee, and S. H. Winoto "Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics" Vol. 09 pp. 49-61 (2003)
6. Mohammed Khudhair Abbas, "Cavitation in centrifugal pumps", "Diyala Journal of Engineering Sciences", pp. 170-180, (2010).
7. D. Somashekar and Dr. H. R. Purushothama, "Numerical Simulation of Cavitation Inception on Radial Flow Pump", "IOSR Journal of Mechanical and Civil Engineering", Vol. 1, Issue 5, pp. 21-26, (2012).
8. Erik Dick, Jan Vierendeels, Sven Serbruyns and John Vande Voorde, "Performance prediction of centrifugal pumps with cfd-tools," Task Quarterly 5, no 4, 579–594, (2001).
9. S R Shah, S V Jain and V J Lakhera, "CFD based flow analysis of centrifugal pump", "Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power", IIT Madras, Chennai, (2010).
10. S.Rajendran and Dr.K.Purushothaman, "Analysis of a centrifugal pump impeller using ANSYS-CFX", "International Journal of Engineering Research & Technology", Vol. 1, Issue 3, (2012).