



IJRASET

International Journal For Research in
Applied Science and Engineering Technology



INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 10 **Issue:** XII **Month of publication:** December 2022

DOI: <https://doi.org/10.22214/ijraset.2022.47958>

www.ijraset.com

Call:  08813907089

E-mail ID: ijraset@gmail.com

Design and CFD Analysis of Centrifugal Pump

Prathamesh Raut¹, Rohit Rathod², Rohit Tidke³, Niraj Rathod⁴, Sanchitee Rokade⁵, Prof. Nishant Kulkarni⁶
^{1, 2, 3, 4, 5, 6}Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune

Abstract: The purpose of this report is to identify /observe and determine the pattern of velocity profile and pressure distribution by using CFD simulation program after the 3D design and modeling of the pump is made using Vista CPD. We have also created a Solid model using Fusion 360 to get a clear idea of Centrifugal pump design. Basically, this report revolves around the idea of investigating the effect and distribution of velocity profile and pressure within a pump having the following specification, Head = 20 m, Flow rate = 100 m³/hr, and RPM = 2000. 3D Navier–Stokes equations were solved using ANSYS CFX. The standard k –ε turbulence model was chosen for the turbulence model. From the design point of view, we have studied the effects of different parameters like rotational speed, volume flow rate etc on the impeller and volute. From the simulation results it was observed that the pressure increases gradually from impeller inlet to outlet. The static pressure on the pressure side is evidently larger than that on the suction side at the same impeller radius. In addition to this, it was observed that the velocity increases from the impeller inlet until it enters the volute casing. It then drops to a minimum value at the outlet region.

Keywords: Centrifugal pump design, Fusion 360, CFD Analysis, Simulation, ANSYS CFX, Vista CPD, pressure distribution, CFD-Tool.

I. INTRODUCTION

Centrifugal pumps which belong to a wider group of fluid machines called turbo machines are the most common type of pump used to move liquids through a piping system. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward or axially into a diffuser or volute chamber, from where it exits into the downstream piping system. Centrifugal pumps are typically used for large discharge through smaller heads.

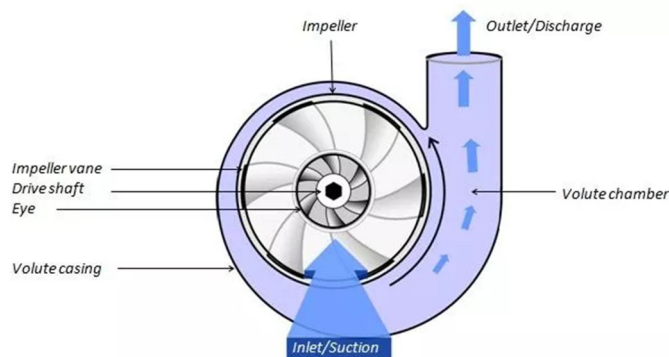


Fig 1: Liquid flow path inside a centrifugal pump

Computational fluid dynamics (CFD) analysis is being increasingly applied in the design of centrifugal pumps. With the aid of the CFD approach, the complex internal flows in water pump impellers, which are not fully understood yet, can be well predicted, to speed up the pump design procedure. Thus, CFD is an important tool for pump designers. The use of CFD tools in the turbo machinery industry is quite common today. Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labour intensive, reducing time and, hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world.

1) FUSION 360 – is one of the widely used CAD software by design engineers across the globe. It's the first 3D CAD, CAM, and CAE tool of its kind, connecting your entire product development process into one cloud-based platform. It helps to understand the designing aspect of components and machines. Therefore, we have used FUSION 360 to understand the designing parts of the Centrifugal Pump.

2) ANSYS Turbo system – V2021/R2 which is one of the CFD tools offers a complete suite of software tools for comprehensive turbomachinery design and analysis. This system will provide a streamlined workflow using Integrated, easy to use environment for all engineering simulations /Analysis using Vista TF, FLUENT, ANSYS FEA and CFX. In a CFD model, the region of interest, a pump casing for example, is subdivided into a large number of cells which form the grid or mesh. In each of these cells, of which there may typically be 300,000, the PDEs can be rewritten as algebraic equations that relate the velocity, pressure, temperature, etc. in that cell to those in all of its immediate neighbors. The resulting set of equations can then be solved iteratively, yielding a complete description of the flow throughout the domain. Powerful graphical post-processors then display the results in an easily understandable way. Therefore, in this report 3D CFD analysis system using Vist CPD together with CFX code is used to simulate the fluid flow through a pump

II. CENTRIFUGAL PUMP DESIGN

The design of centrifugal pump is divided in two categories: Impeller Design and Volute Design. The detailed procedure of single volute casing and impeller design can be found in different literature; in this report vista CPD for the design of centrifugal pump is used. The duty parameters required by the pump are assumed to be: 1. Head = 20 m, 2. Flow rate = 100 m³/hr, 3. RPM = 2000, 4. Density = 1000 Kg/ m³,

A. Solid Modeling Of Centrifugal Pump In Fusion 360

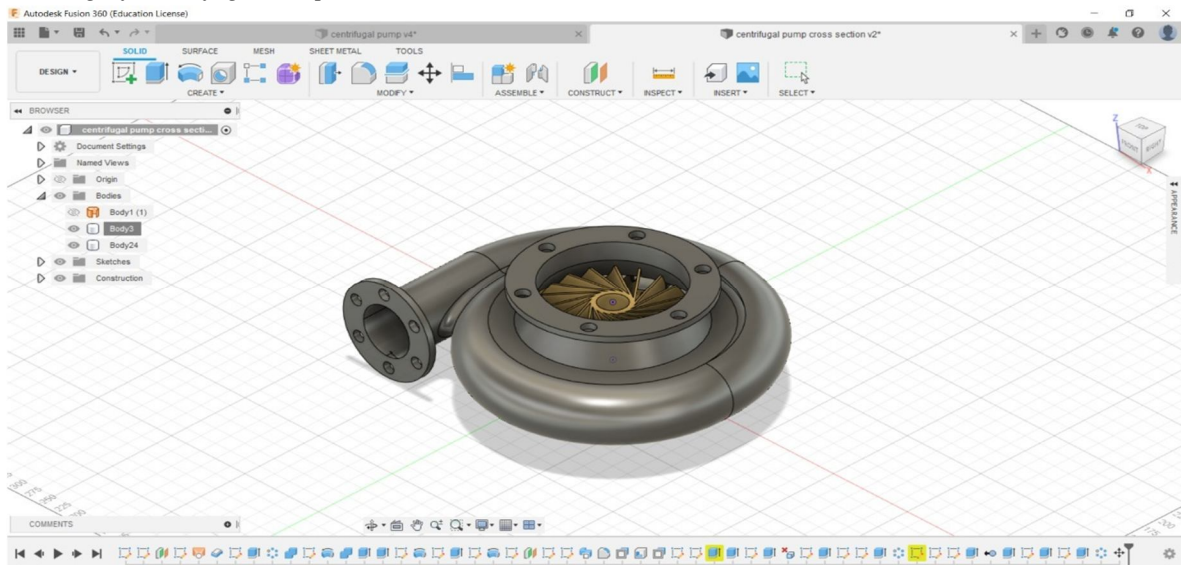


Fig. 1: 3D Model of Centrifugal Pump made in Fusion 360

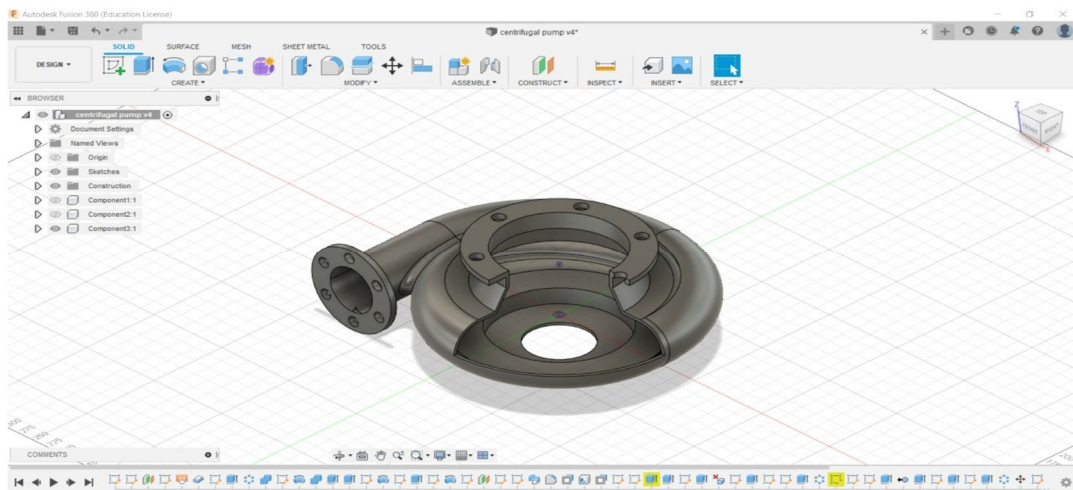


Fig. 2: Design Geometry of Volute

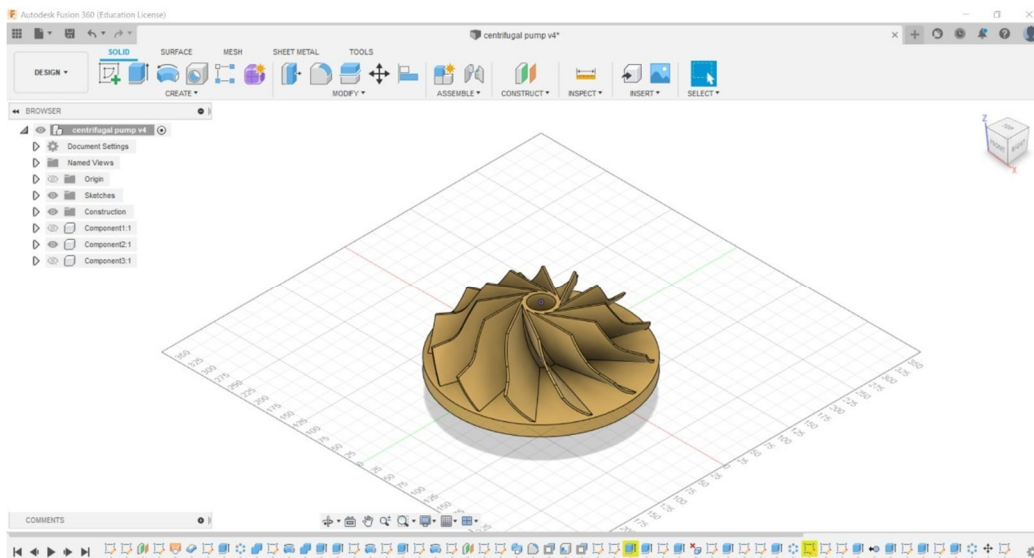


Fig. 3: Design Geometry of Impeller

B. Impeller Design using Vista TF V2021/R2

Input variables are used to give a basic starting point for the pump design. The head, volume flow, rotational speed and other parameters could be changed to the specific purpose. Various windows show the design parameters, like the angle and thickness distribution. The following fig-2 will demonstrate the entire workflow from input values in Vista CPD to final results by Vista design module. This way, manipulation of the geometry in BladeGen or BladeEditor will be possible and all the next steps will be automatically generated and results produced.

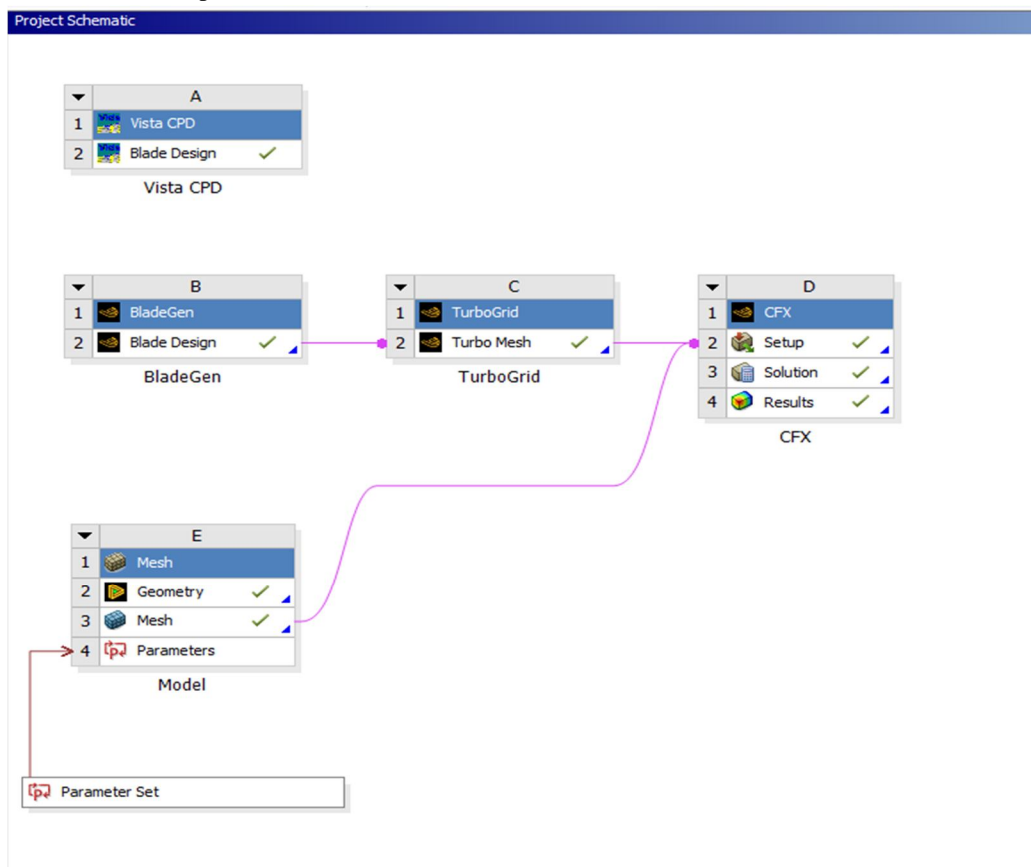


Fig. 4: Flow chart of the project.

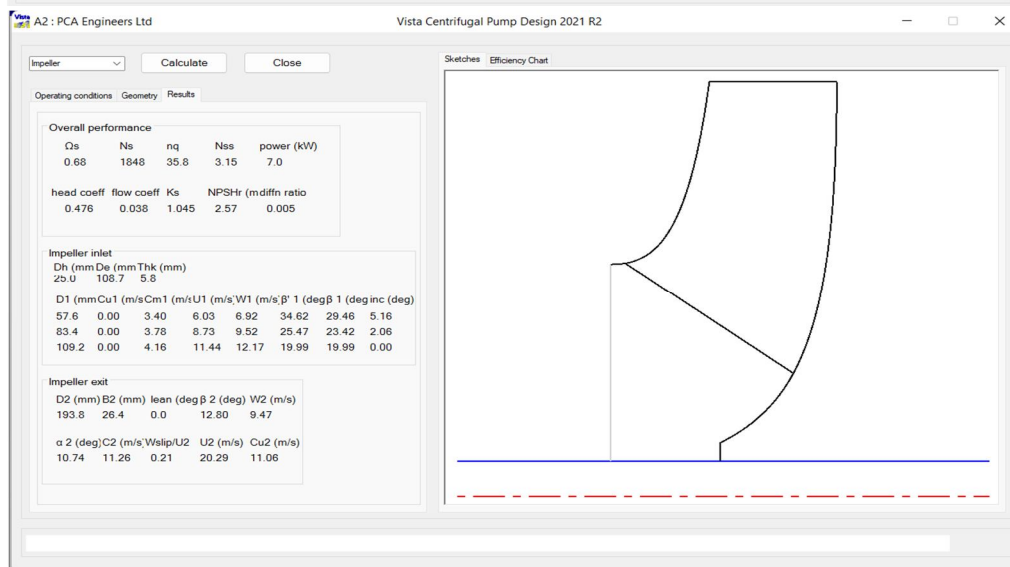
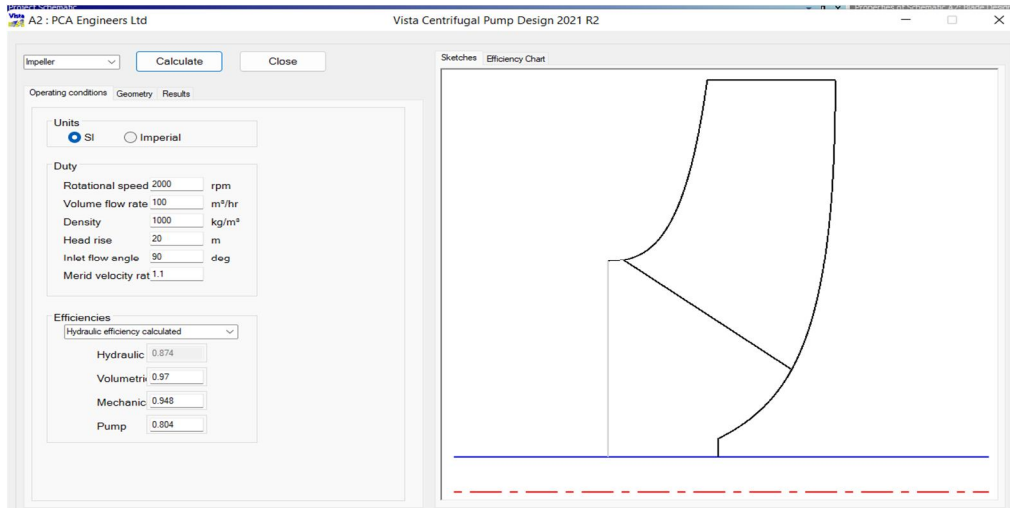


Fig.5: Input design parameters of pump using Vista CPD software.

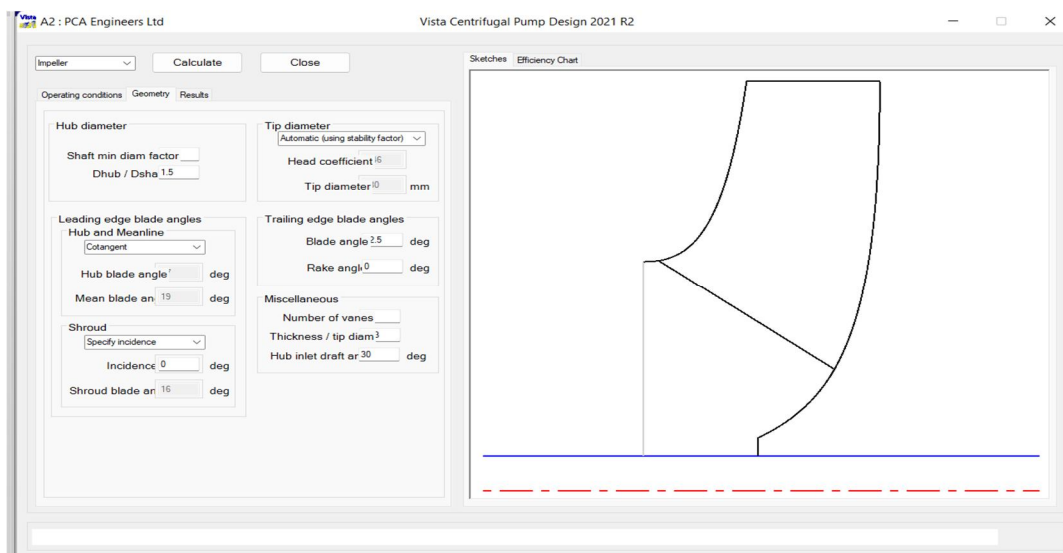


Fig.6: Geometrical parameters for Impeller of the pump using Vista CPD software

C. Volute Design

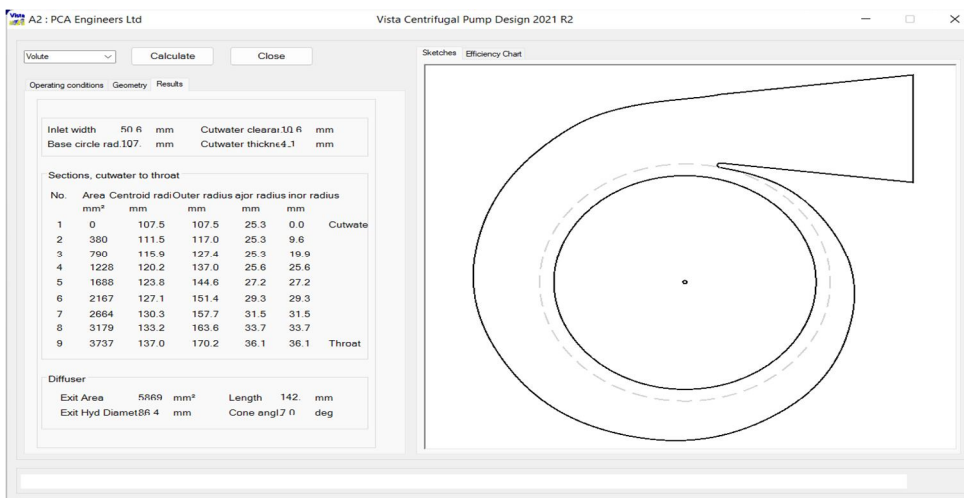
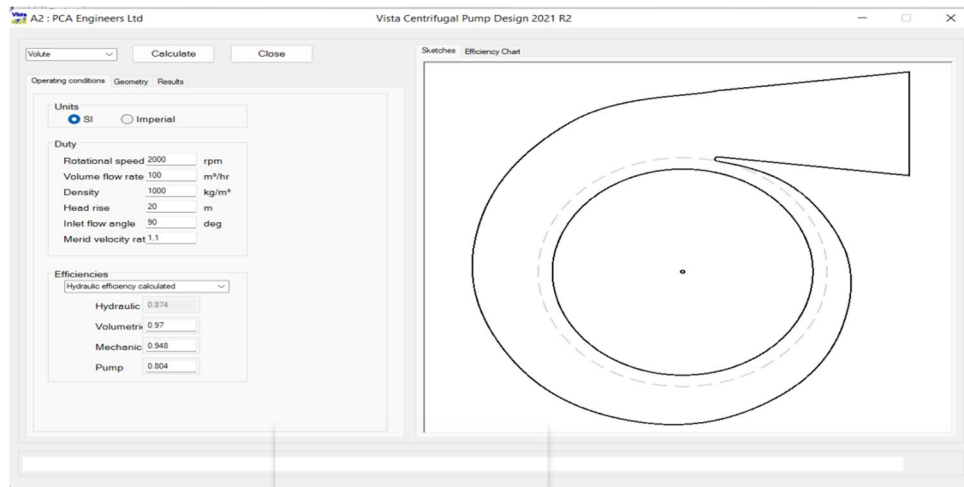


Fig.7: Output parameters using Vista CPD software.

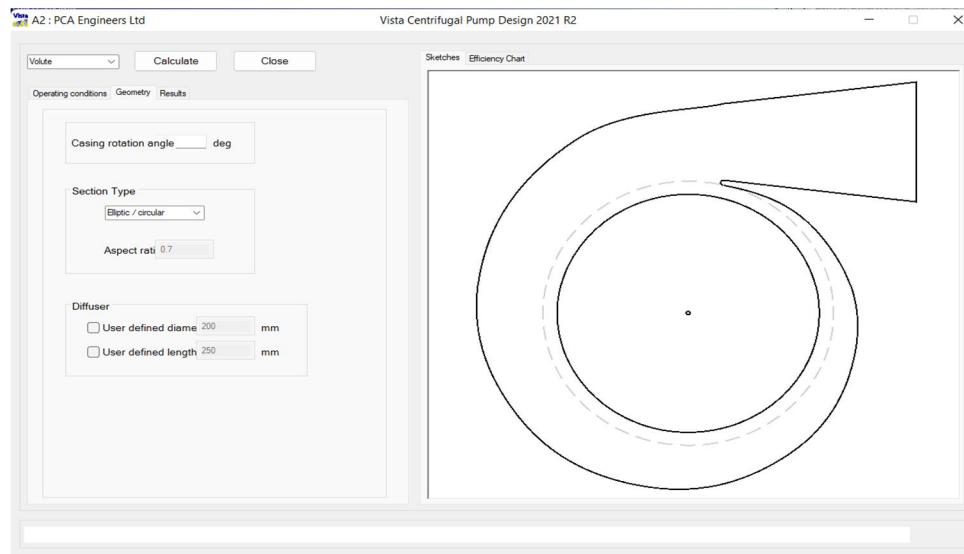


Fig.8: Geometrical parameters for volute using Vista CPD software.

III. MESH GENERATION

Once the pump geometry has been specified and a mesh has been created automatically, where the flow equations need to be solved.

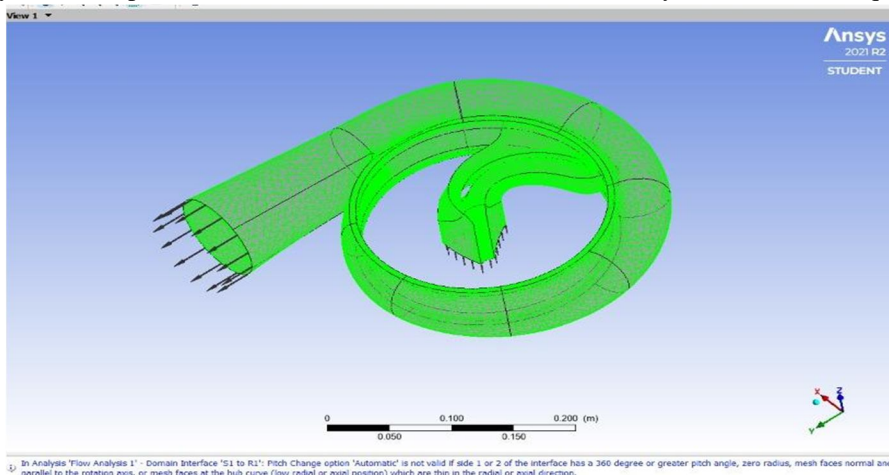


Fig.9: Automatic mesh profile for centrifugal pump.

IV. ANALYSIS SETUP

Design points for a parametric study can be specified using the required duty of the pump in the setup steps:

- 1) *Input Material:* Material is also assigned to the parts of the pump as: Casing and Impeller: Aluminum alloy, Hydraulic Region: Water, Rotating part: Rotating region
- 2) *Boundary Conditions:* Boundary conditions are applied to the inlet and outlet of the pump i.e., 0 pa at inlet, 100 m³/hr at outlet, and 2000 RPM.

A. Solution Initialization

Initialization in Ansys CFX is done by providing initial guess values to solve the governing equation so that the flow field variables can be solved by iteration toward the solution. The default automatic initialization for the velocity and static pressure is used to provide a start point to the solution.

V. CFD RESULTS

After analysis has been carried out the following results are obtained. The results are taken only when the convergence is obtained for the solution. As the solution iterated 1000 times and the pump impeller completed a full turn, following results are taken from different axis and cross-sections.

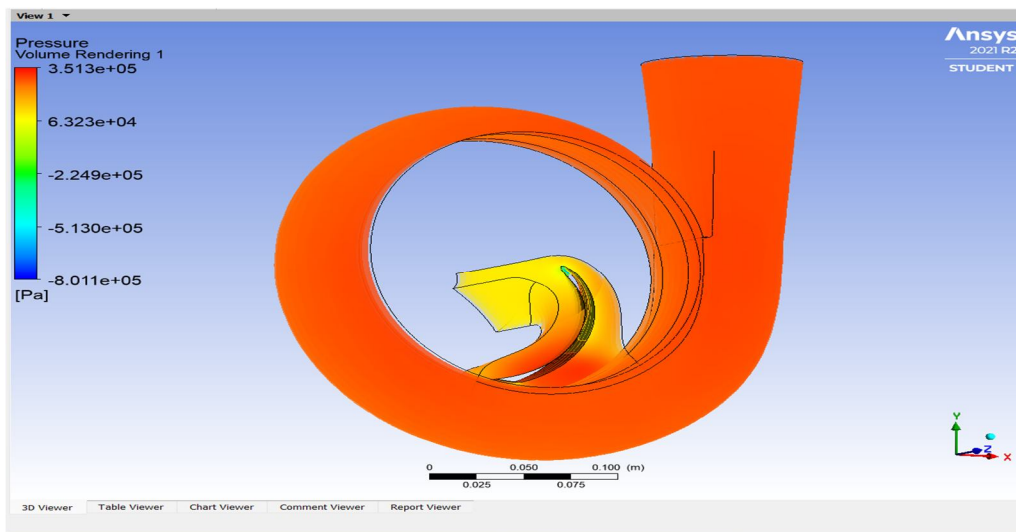


Fig. 10: Pressure Volume Rendering

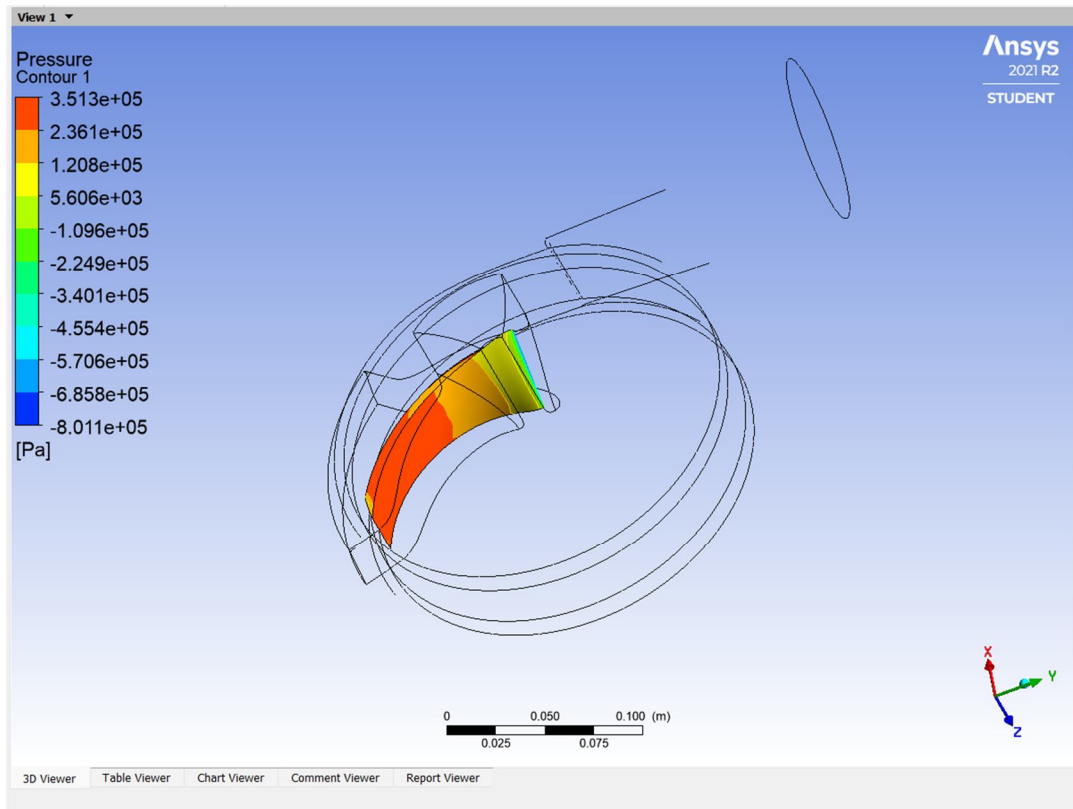


Fig.11: Pressure Contour

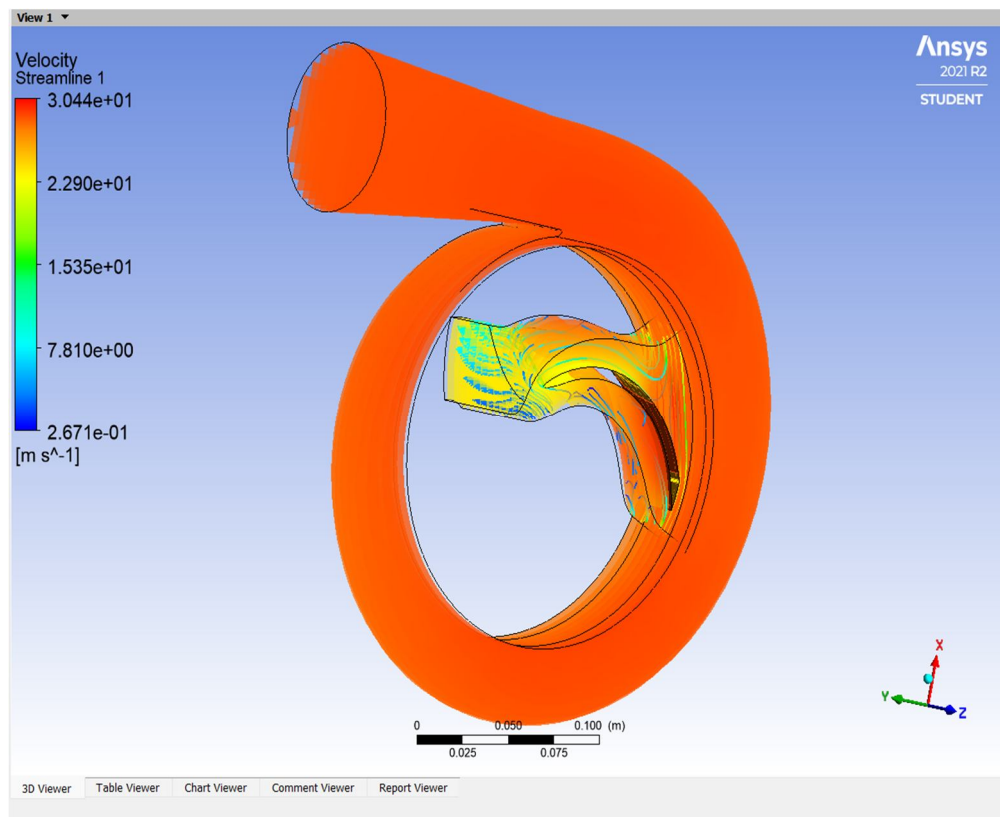


Fig.12: Velocity Streamline

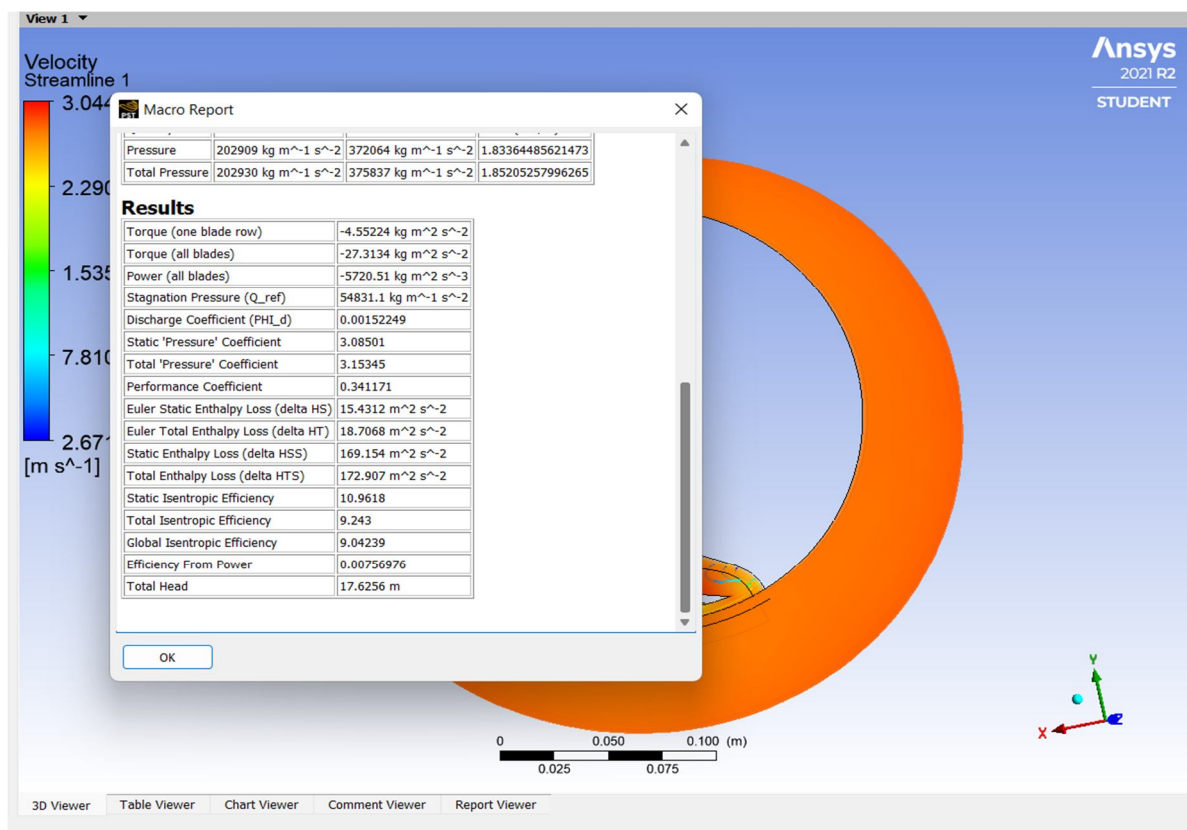
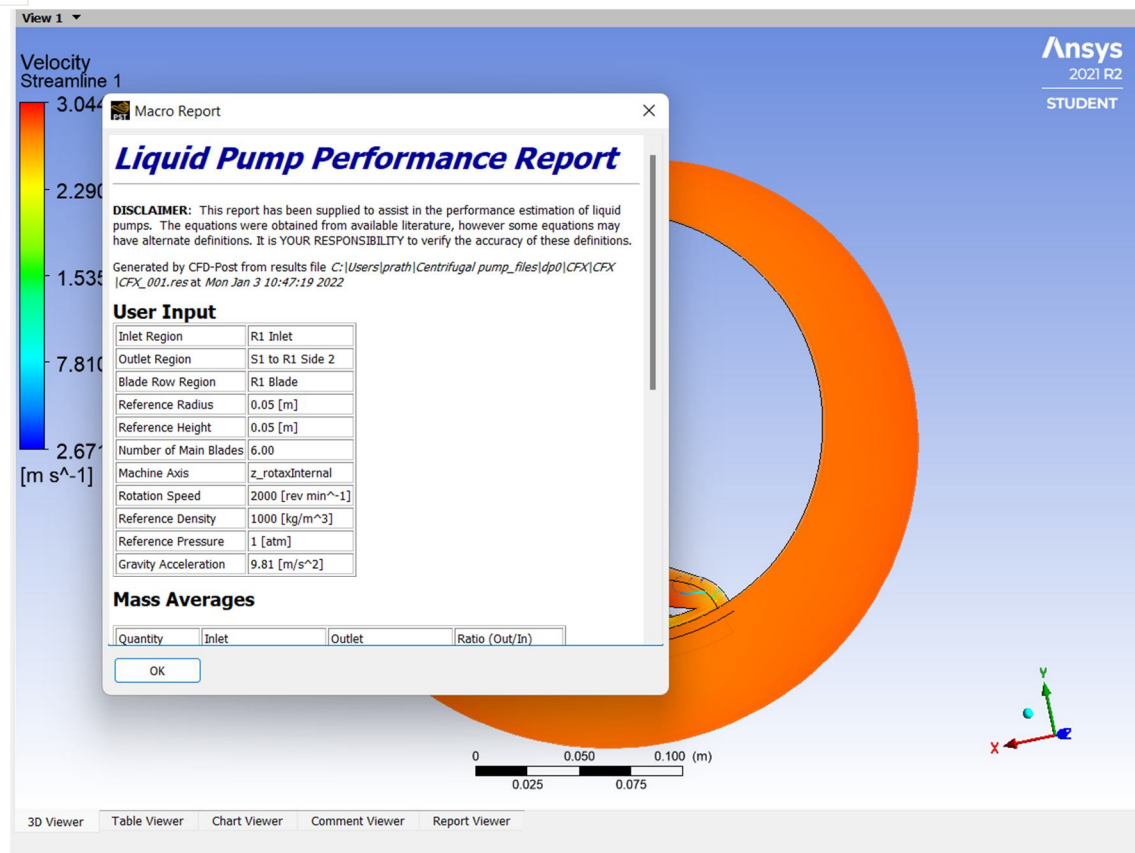


Fig.9: Pump Performance Report

VI. CONCLUSION

Experimental methods and past experience are undoubtedly important, but the most effective way to study pump performance is through Computational Fluid Dynamics (CFD). And with the help of a solid model which we made using Fusion 360 we were able to study the aspects of designing a centrifugal pump.

The CFD-code (ANSYS Turbo system – 2021/R2 (Ansys CFX), version 2021/R2), has been used in this report for the flow analysis of pump with end-suction volute type: The impeller and volute geometry was designed by Vista TF CPD V2021/R2 software by assuming the required duty parameters by the pump to be design as a case study are: Head = 20 m, Flow rate = 100 m³/hr, RPM = 2000, Density = 1000 Kg/ m³, and the model prepared has been analyzed in CFD tool CFX and its performance is analyzed at different flow rates. It is found that the design and analysis methods lead to completely very good flow field predictions. This makes the methods useful for general performance prediction. In this way, the design can be optimized to give reduced energy consumption, lower head loss, prolonged component life and better flexibility of the system, before the prototype is even built.

REFERENCES

- [1] 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.
- [2] 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.
- [3] P.UshaShriansC.Syamsundar, "computational analysis on performance of a centrifugal pump impeller," Pro
- [4] E.C. Bacharoudis, A.E. Filios, 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, no 2, 75-83, 2008.
- [5] ANSYS CFX basic information (<http://www.ansys.com/Products/Simulation+Technology/Fluid+Dynamics/Fluid+Dynamics+Products/ANSYS+CFX>) ANSYS CFX technical specification (http://www.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/brochure/ansys-cfx-tech_specs.pdf)



10.22214/IJRASET



45.98



IMPACT FACTOR:
7.129



IMPACT FACTOR:
7.429



INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089  (24*7 Support on Whatsapp)