Design and Analysis on a Semi open Impeller Blades

GOWTHAM VARMA BHUPATHIRAJU [1], S RAMESH KUMAR [2]

[1]UG Students, Department of Mechanical Engineering KL University, Guntur Dist A.P, India

[1]Researcher, FIPPLE IT TECHNOLOGIES PRIVATE LIMITED, Karnavanipalem, Visakhapatnam, Andhra Pradesh, India.

[2]Assistant Professor, Department of Mechanical Engineering, KL University, Guntur Dist A.P, India.

[1]gowthamvarma810@gmail.com

[2] cadramesh@kluniversity.in

Abstract — *The objective of this paper is to model and carryout the fluid analysis on impeller which is used in centrifugal pump to increase its power and efficiency of the veins. In this work impeller was designed with Solid works and analysis is done in ANSYS CFX. The first stage involves the modeling using reverse engineering technique, the second stage involves in mesh generation and refinement on domain of the designed impeller, The third stage deals with the identification of initial and boundary conditions of the mesh-equipped module and at last stage, the impeller is analyzed to obtains various results for identifying the factors that affecting the impeller performance.*

Keywords (Size 10 & Bold) — *Solid works, ANSYS CFX. Reverse Engineering, Mesh-Equipped.*

I. Introduction

In the present day's the pump industry and other technical sectors, have a great deal of getting maximum efficiency on the impeller with in their design performance, the impeller is a mono block component of the centrifugal pump [1], which transfers energy from the motor that drives the pump and then the fluid is pumped by accelerating the fluid outwards from the center of rotation. It is a great deal of labour and facility will be saved, as well as it helps in shortening the design cycle. They also require a high level of reliability and efficiency in order to be cost-effective. The design performance prediction process is still a difficult task, due to the great number of free geometric parameters, which cannot be directly evaluated. The significant cost and time of the process by constructing of prototypes reduces the profit of the pump manufactures, for that reason modelling and fluid analysis are being done with various pumps. Therefore, great improvement on centrifugal pump design must be achieved by CFD analysis of inner flow inside a centrifugal pump and following application of its results in pump design processes. For this reason CFD analysis is currently being used in the design and construction stage of various pump types. And challenge of improving the hydraulic efficiency requires the reverse engineering techniques. The process of duplicating an existing part, subassembly, or product, without drawings, documents, or a computer model is known as reverse engineering. The mesh generation process is a laborious task for many CFD codes, and the quality of the final mesh depends considerably on the user's experience. An alternative practice in complex domains is the use of Cartesian grids that require a much reduced construction effort. However such grids cannot be everywhere body-fitted, and for this reason various numerical techniques have been developed to improve the accuracy in these regions. A Cartesian mesh approach is also followed in the present work, where an advanced numerical technique is incorporated in order to eliminate the grid generation cost and to represent with adequate accuracy the complex geometry of a centrifugal pump impeller [2]. The latter is parameterized using a reduced number of controlling geometric variables, facilitating the investigation of their individual or combined effects on the flow and the impeller performance.

II. Modeling Phase

In the modelling phase at first the dimensions of the impeller is taken from the pattern of the impeller then with those dimensions reverse engineering technique is applied to that such as which are drawn in solid works and made into a solid model with the help of those dimensions [3].

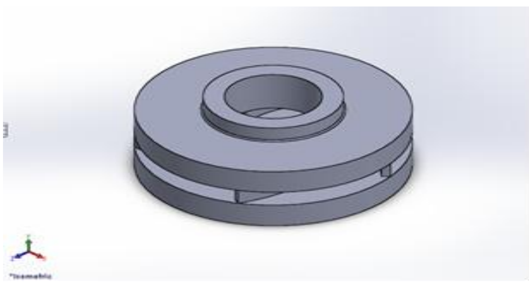

Fig 1. Model of the impeller in Solid works

Table 1.Specification of the impeller

Specification of the impeller	
Parameter	Dimensions
Inlet Diameter (cm)	137.32
Outlet Diameter (cm)	48.50
Thickness of the blade (cm)	4.5
Height of the blade (cm)	10
Number of blades	5

III. Analysis phase

In the Analysis phase there undergoes few steps such as mesh generation, refinement on domain of the designed impeller, identification of initial and boundary conditions of the mesh-equipped module and finally the impeller is analysed to obtain various results for identifying the factors that affecting the impeller performance.

- **Mesh generation**

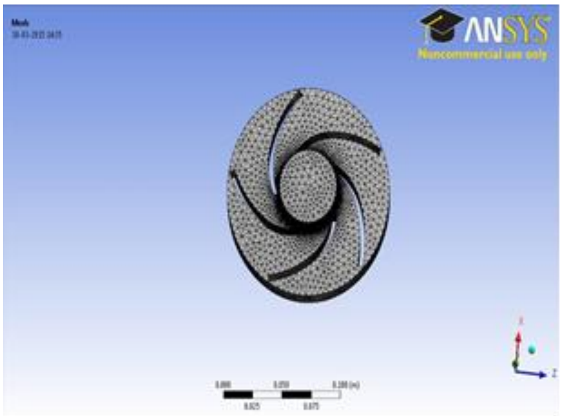

Fig 2.Mesh generation of the impeller in ANSYS CFX

Mesh generation is one of the most critical aspects of engineering simulation. Too many cells may result in long solver runs, and too few may lead to inaccurate results. ANSYS Meshing technology provides a means to balance these requirements and obtain the right mesh for each simulation in the most automated way possible. ANSYS Meshing technology has been built on the strengths of stand-alone, class-leading meshing tools. The strongest aspects of these separate tools have been brought together in a single environment to produce some of the most powerful meshing available.

- **Identification of Initial and boundary conditions**

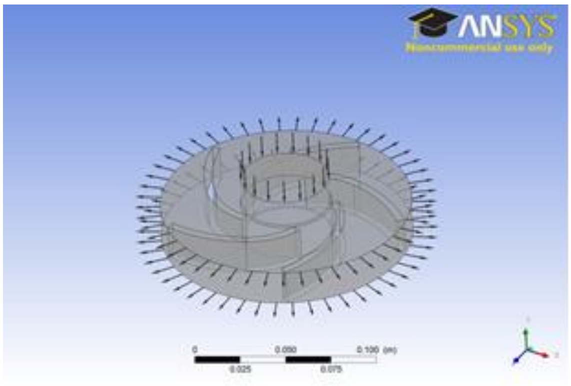

Fig 3.Boundary conditions of the impeller in ANSYS CFX

In this stage Boundary conditions are defined to the model with the parameters like inlet, outlet, walls, rotatory, hub, blades and stationary objects and thing that are required to solve the equations are given.

Procedure in the analysis stage

In the Analysis stage the analysis is done using ANSYS CFX where at first the model is imported into geometry in that geometry will be modified if necessary like adding or subtraction of the surfaces like fluid regions and solid regions Booleans.

After that it is updated and meshing of the impeller takes places, when the mesh gets generated, after that setup opens and will give necessary things in that like input, output, fixed, rotatory etc. values are given and before that name sections are created to them, and in the solution stage which is made to run with the iterations in the solution. When the solution run results are obtained [4].

IV. Parameters considered during analysis

Table 2

1	Material type	Water
2	Reference pressure	0 atm
3	Angular Velocity	2500 radian/min
4	Relative Pressure	700000 pa
5	Outlet mass flow rate	35 kg/s
6	Mass of the fluid	1000 kg/m^3
7	Temperature	25 c

V. Results

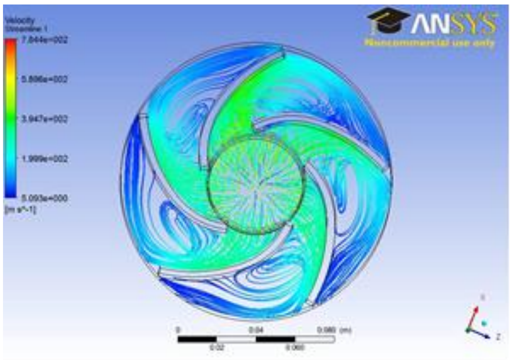

Fig 4. Stream line flow velocity

This is the stream line flow which takes from the inlet to the outlet with in the range of -5.093e+000 to +7.844e+002 (m s^-1).

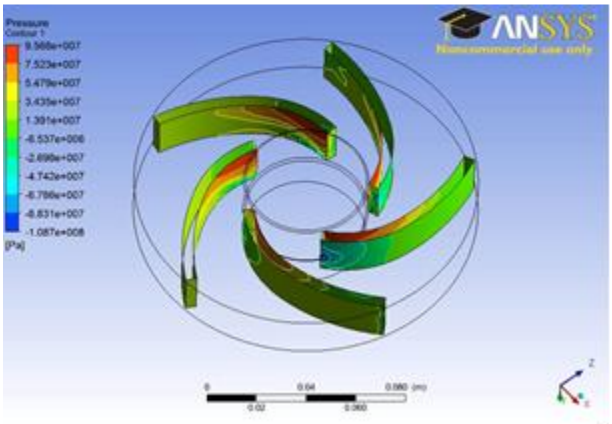

Fig 5. Pressure contour

This is pressure contour on the blades when the fluid flows on to the surfaces of the blades with in the range of -1.087e+008 to +9.568e+007 (Pa).

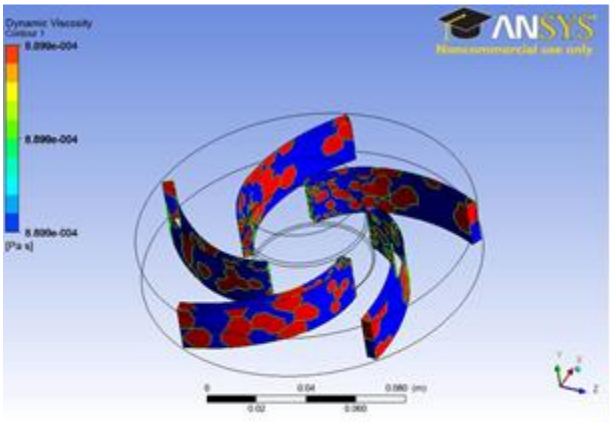

Fig 6. Dynamic viscosity

This is Dynamic viscosity where the fluid flow on to the surface of the blades which ranges from -8.899e-004 to +8.899e-004 (Pa s).

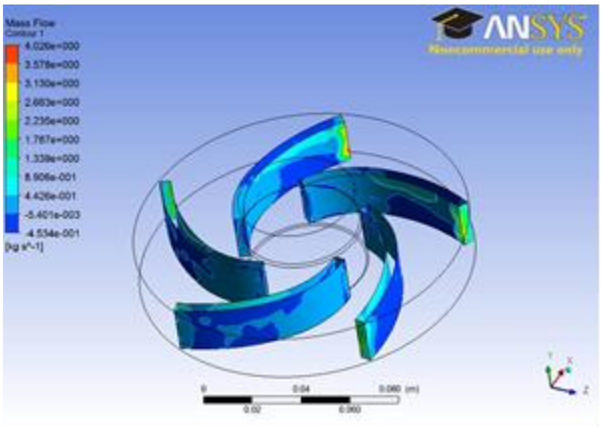

Fig 7. Mass flow contour

This is mass flow rate of the fluid when the it touches the surface of the blades with in the range to -4.534e+001 to +4.026e+000 (kg s^-1).

VI. Future Scope

To change the dimensions of the materials and even can change the angle of the veins of the impeller to get the better efficiency.

VII. Conclusion

The impeller is modelled in Solid works using Reverse engineering techniques. The flow through a centrifugal pump of the impeller was analysed using commercial CFD package ANSYS-CFX. CFD analysis was carried out at design and off design condition and is reported.

The Simulation was performed by using turbulent modelling Shear stress Transport with automatic wall function.

Performance charts, cavitation analysis, pressure contours and velocity vector contour are predicted.

- The mesh is generated successfully using ANSYS-CFX. The performance results are satisfactorily matching with test data, hence mesh quality is good.
- The increase of the designed flow rate causes a reduction in the total head of the pump.

VIII. References

[1] http://en.wikipedia.org/wiki/Centrifugal_pump

[2] http://en.wikipedia.org/wiki/Impeller

[3] A TextBook of Fluid Mechanics and Hydraulic Machines - Dr. R. K. Bansal

[4] WWW.PUMPFUNDAMENTALS.COM/OPENVSENCLOSEDIMPELLER.PDF