International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 07 | July 2021 www.irjet.net p-ISSN: 2395-0072

Flow Simulation of Centrifugal Pump by Varying Mass Flow Rate Using CFD

Saurabh Sharma[1], Rahul Kumar[2]

[1]*Assistant Professor, Dept. of Mechanical Engineering, Shankara Institute of Technology, Jaipur*
[2]*Assistant Professor, Dept. of Mechanical Engineering, Shankara Institute of Technology, Jaipur*

---***---

Abstract - *The flow simulation of centrifugal pump has been carried out using CFD tool i.e. ANSYS CFX to investigate the effect of different mass flow rate on pump characteristics. As mass flow rate increases, the maximum pressure occur at trailing edge of the blade which is shown in form of red color. Also, as the flow rate increases, the pressure along the trailing edge increases. The maximum value of pressure occurs at higher flow rate. at different flow rates. It is clear that with increase in flow rate, velocity increases. So streamline velocity is uniform in this case. Due to some losses, swirl can be seen and maximum velocity occurs at high flow rate.*

***Key Words*: ANSYS CFX, Computational fluid dynamics, Flow simulation, Centrifugal pump, Impeller, Efficiency, Blades.**

1. INTRODUCTION

Centrifugal pump works on the principle that when fluid is rotated by an external source, it is thrown away from central axis of rotation and a centrifugal head is influenced which enables it to rise to a higher level. They can be used for water treatment plants, sewage, drainage, irrigation, hydraulic power services, and for various purpose in industrial as well as other sectors. Centrifugal pumps are cheaper and easy to install and their maintenance is easy.

1.1 Classification of Pumps

1. Rotodynamic pumps
- Radial flow pumps
- Axial flow pumps
- Mixed flow pumps

2. Positive displacement pumps
- Gear pumps
- Vane pumps
- Piston pumps

In rotodynamic pumps, increase in energy level is due to a combination of centrifugal energy, pressure energy and kinetic energy. The energy transfer takes place in a radial flow pump when flow is in its radial path. In an axial flow pump, the energy transfer occurs when flow is in its axial direction. The energy transfer in a mixed flow pump occurs when the flow takes place in the direction that comprises radial as well as axial components.
The radial flow pumps are commonly called centrifugal pumps.

1.2 Components of Centrifugal Pump

1.2.1 Impeller- It has series of forward or backward curved vanes which is mounted on the shaft which is coupled to an electric motor.

1.2.2 Casing- It is an airtight chamber that surrounds the pump impeller. It is used to guide water to and from the impeller. Water is partially converted from kinetic energy to pressure energy by casing.

1.2.3 Suction Pipe- The pipe which connects the center of the impeller to sump from which liquid is to be lifted is suction pipe. In order to prevent entry of any debris or solid particles into the pump, strainer is provided at its lower end. The foot valve opens in the upward direction and is fitted at lowest end of the pipe.

1.2.4 Delivery Pipe- The pipe which is connected at lower end to the outlet of the pump and delivers the liquid up to a desired height is delivery pipe. In order to regulate the supply of water, a regulating valve is provided on the delivery pipe.

2. Model Geometry

2.1 Geometrical Data

Table 1 Geometrical data of impeller

Impeller inlet: Hub	
diameter, Dh Eye	34.9 mm
diameter, De	150.7 mm
D1	114.7 mm
B1	5.2 mm
Impeller exit: Hub	
D2	260.4 mm
B2	34.1 mm
Trailing edge blade angle	22.5^0
Hub inlet draft angle	3^0
Thickness- tip diameter ratio	0.02
Number of blades	7
Rotational speed	1500 rpm

2.2 Impeller Modelling

Impeller of centrifugal pump has been modeled using ANSYS- Workbench, which is an intuitive up- front tool used in conjunction with CAD systems or Design Modeler. ANSYS Workbench is a software that is used to perform various analysis like structural, thermal or electromagnetic. The software focuses on geometry creation and optimization, attaching the existing geometry, setting up the finite element model, solving, reviewing the results, etc. It also provides an idea about the basic finite element simulation concept as well as result interpretation. So, ANSYS Workbench is the backbone for delivering a comprehensive as well as integrated simulation system, which results in higher productivity from integrated applications and access to Multiphysics and systems level capabilities.

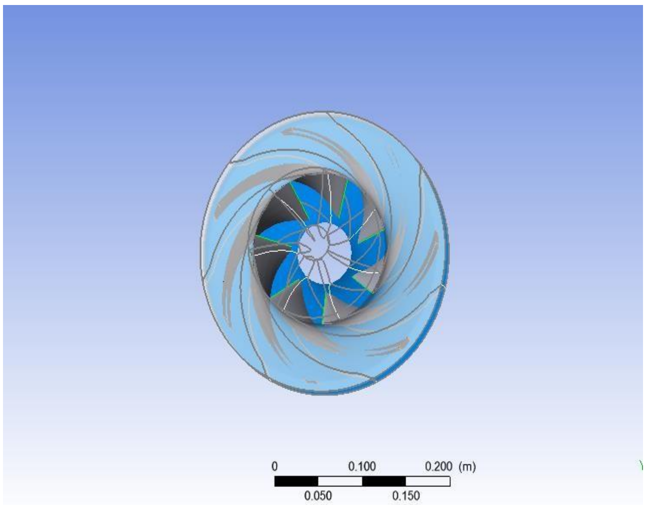

Fig 1 modeling of impeller

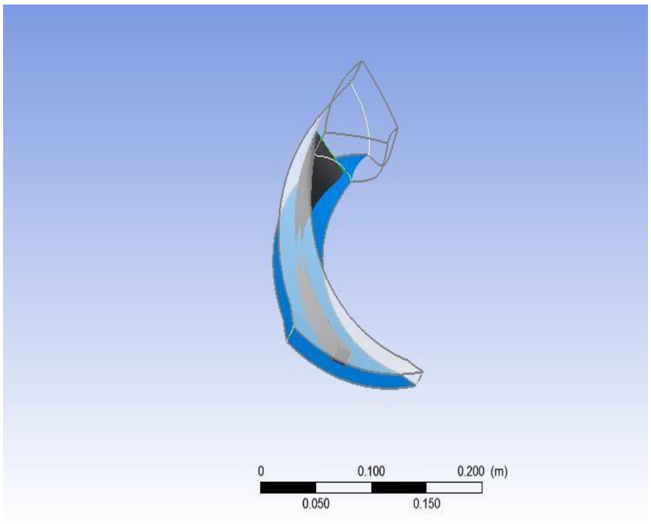

Fig 2 Modeling of single blade

2.3 Mesh Generation of Impeller Using Turbogrid

After the creation of model in first step, it is then exported to ANSYS Turbogrid. This is a software that targets complete automation combined with an unprecedented mesh quality level for most of complex blade shapes. The final mesh size, which is desired, is defined and all other steps are operated automatically to produce a mesh of extremely high quality. In this, the grid angles are good, mesh sizes transition smoothly, and high aspect ratio elements are generated near- wall regions in order to resolve these regions efficiently and capture boundary layer flows accurately. The impeller meshing of 7 blades using Turbogrid has been shown.

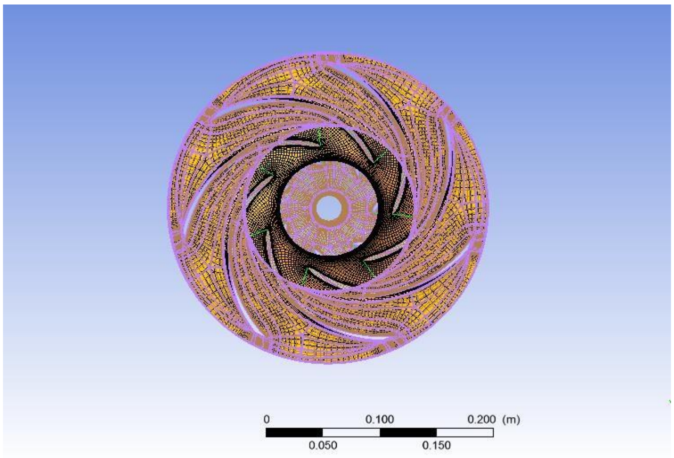

Fig 3 Meshing of impeller with 7 blades

Table 2 Mesh statistics of impeller with 7 blades

Mesh Type	Structural or Regular
Number of Nodes	83622
Number of Elements	74080

Elements Type	Hexahedral

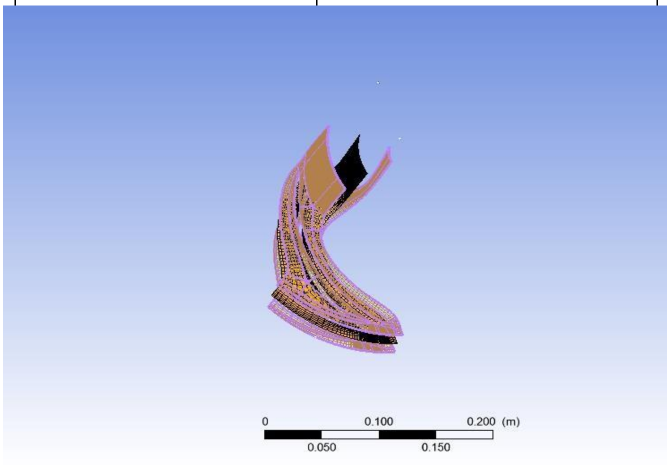

Fig 4 Meshing of single blade

Table 3 Meshing of single blade

Mesh Type	Structural or Regular
Number of Nodes	83622
Number of Elements	74080

2.4 Modeling of Casing

Table 4 Geometrical data for casing

Section type	Circular
Aspect ratio	0.7
Diffuser diameter	200mm
Diffuser length	250mm
Casing rotation angle	14
Inlet width	66.6mm
Base circle radius	144.4mm

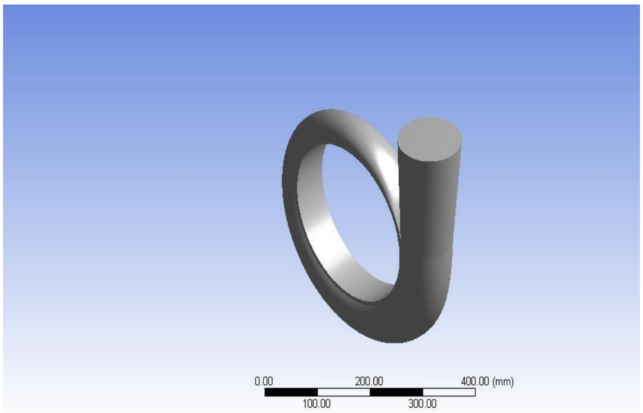

Fig 5 Modeling of casing

2.5 Mesh Generation of Casing

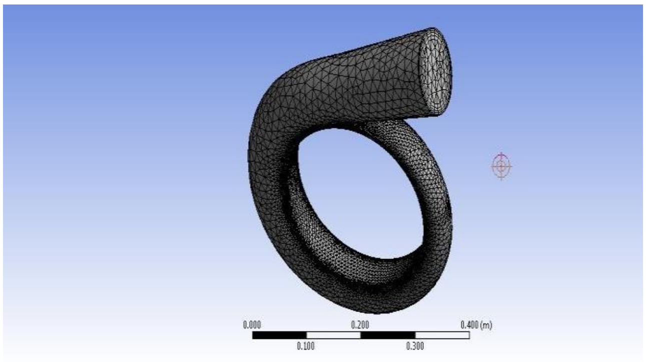

Fig 6 Meshing of casing

Table 5 Mesh statistics of casing

Mesh Type	Un-structural or irregular
Number of Nodes	64038
Number of Elements	184009
Element type	Tetrahedral

3. Boundary Conditions

After the development of meshing, the mesh model is exported to the CFX- Pre, a preprocessing stage. At this stage, initial conditions like boundary conditions, Domain physics, running conditions, etc. is provided to the problem. Following input conditions are provided as input to the pre- processor.

3.1 Domain Physics for Impeller

Table 6 Domain physics for impeller

Location	Entire Passage
Type of Fluid	Water
Turbulence Model Used	Shear Stress Transport (SST)
Flow Direction	Normal to Boundary
Reference Pressure	0 [atm]
Static Pressure	1 [atm]

Table 7 Boundary condition templates

Inflow boundary condition	Mass flow inlet
Outflow boundary condition	P- static outlet
Mass flow	Per component
Mass flow rate	55.5 kg/sec, 61.11 kg/sec, 66.67kg/sec, 72.2 kg/sec
Wall roughness	100 µm
Wall influence on flow	No Slip
Turbulence intensity	1%

In this analysis, mass flow rate is used at inlet and static pressure at outlet. Inner walls are subjected to be rotational (impeller) while the outer walls are stationary (volute). There are interface between stationary as well as rotational regions. No slip boundary conditions are imposed over the impeller blades and walls. The surface roughness is considered to be 100- µm and turbulence intensity for all conditions is considered to be 1%. Water is used as working fluid in an ambient condition and in order to obtain more accurate result, Shear Stress Transport (SST) turbulence model has been used.

4. Result and Discussion

Result obtained during the analysis of pump has been obtained in the form of pressure contour and velocity streamline.

4.1 Pressure Distribution and Velocity Distribution for Impeller with 7 Blades for Different Mass Flow Rates

4.1.1 For mass flow rate of 55.5 kg/sec

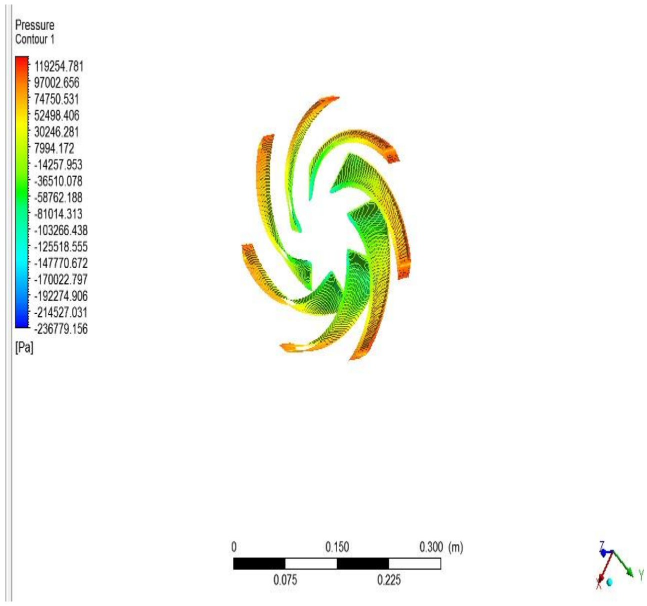

Fig 7 pressure distribution for blade 7 and mass flow rate 55.5 kg/sec

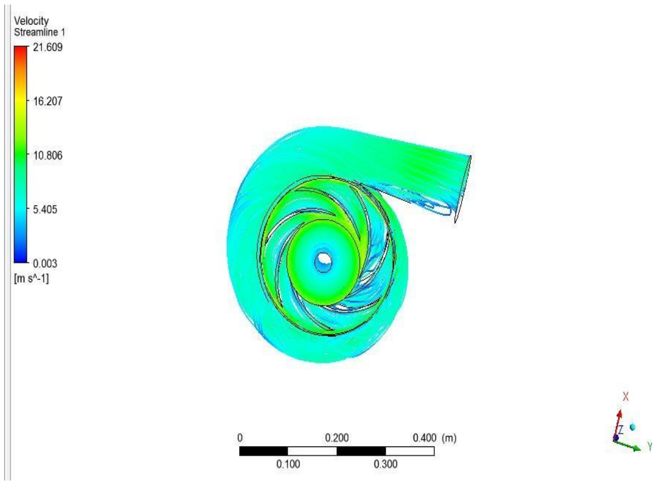

Fig 8 velocity distribution for blade 7 and mass flow rate 55.5 kg/sec

4.1.2 For mass flow rate of 61.1 kg/sec

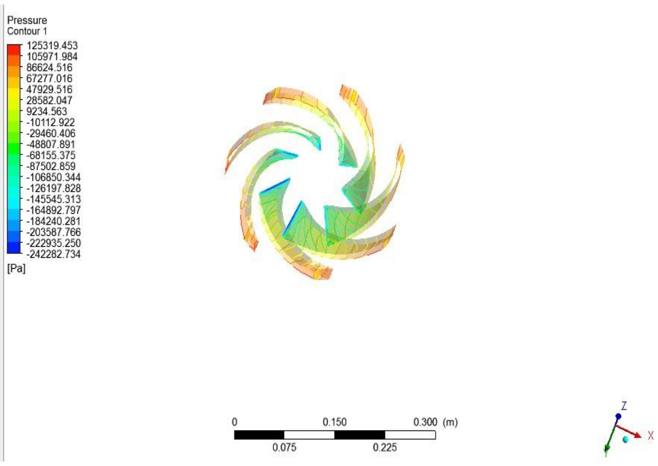

Fig 9 pressure distribution for blade 7 and mass flow rate 61.1 kg/sec

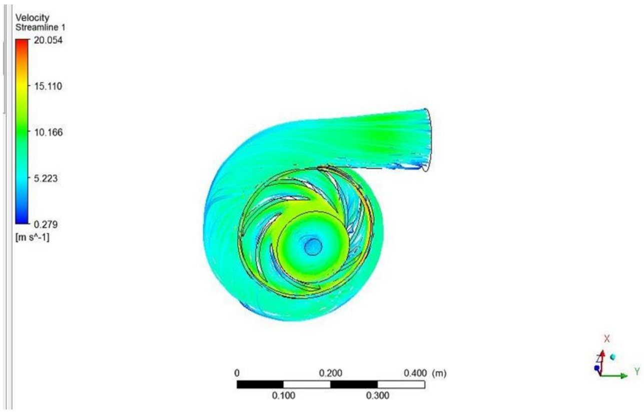

Fig 10 velocity distribution for blade 7 and mass flow rate 61.1 kg/sec

4.1.3 For mass flow rate of 66.67 kg/sec

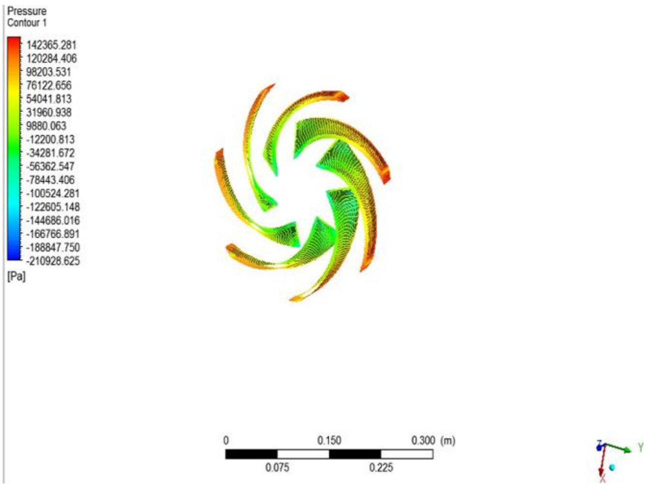

Fig 11 pressure distribution for blade 7 and mass flow rate 66.67 kg/sec

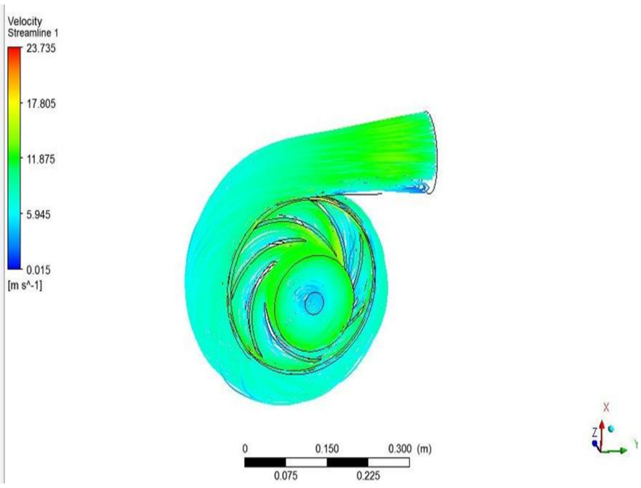

Fig 12 velocity distribution for blade 7 and mass flow rate 66.67 kg/sec

The total pressure distribution has been shown in fig. 7, 9, 11 at different flow rates. From the fig. it is clear that the maximum pressure occurs at trailing edge of the blade which is shown in form of red color. Also, as the flow rate increases, the pressure along the trailing edge increases. The maximum value of pressure occurs at higher flow rate. The velocity distribution has been shown in fig. 8, 10, 12 at different flow rates. It is clear that with increase in flow rate, velocity increases. So streamline velocity is uniform in this case. Due to some losses, swirl can be seen in the fig. Maximum velocity occurs at high flow rate.

5. Conclusions

The flow simulation of centrifugal pump has been carried out using CFD tool i.e. ANSYS CFX to investigate the effect of different flow rates on centrifugal pump characteristics. Following conclusions are drawn from the study:

- The maximum pressure occurs at trailing edge of the blade which is shown in form of red color.
- Also, as the flow rate increases, the pressure along the trailing edge increases. The maximum value of pressure occurs at higher flow rate. at different flow rates.
- It is clear that with increase in flow rate, velocity increases. So streamline velocity is uniform in this case.
- Due to some losses, swirl can be seen and maximum velocity occurs at high flow rate.

References

[1] Anagnostopoulos, CFD Analysis and Design Effects in a Radial Pump Impeller, WSEAS Transactions on Fluid Mechanics 1 (2006) p. 763.M. Young, The Technical Writer's Handbook. Mill Valley, CA: University Science, 1989.

[2] Bacharoudis, E. Filios, A. Mentzos, M. Margaris, Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle, The open Mechanical Engineering Journal 2. (2008) p. 75.K. Elissa, "Title of paper if known," unpublished.

[3] B. Jafarzadeh, A. Hajari, M.M. Alishahi, M.H. Akbari, The Flow Simulation of Low Specific Speed High-Speed Centrifugal Pump, Applied Mathematical Modelling 35(2011) pp. 242-249.

[4] Gamal R.H. Abo Elyamin, Magdy A. Bassily, Khalil Y. Khalil, Mohamed Sh. Gomaa, Effect of impeller blades number on the performance of a centrifugal pump, Alexandria Engineering Journal (2019).

[5] J.S. Anagnostopoulos, Numerical calculation of the flow in a centrifugal pump impeller using Cartesian grid, in: Proceedings of the second WSEAS International conference on Applied and Theoratical Mechanics, Venice, Italy, November (2006).

[6] K.M. Guleren, A. Pinarbasi, Numerical simulation of the stalled flow within a vaned centrifugal pump, J. Mech. Eng. Sci. 218 (2004) pp. 425-435.