International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 3.45 (SJIF-2015), e-ISSN: 2455-2585
Volume 4, Issue 5, May-2018

Computational Fluid Dynamic Analysis of Centrifugal Pump

Disha Dewangan[1]
[1] *Department of Industrial & Production Engg., Institute of Technology Guru Ghasidas Vishwavidyalaya*
Dewangan.disha@gmail.com[1]

Abstract*—Present work deals with the pattern of velocity profile and pressure distribution using CFD simulation and later on the 3D design and modelling of the centrifugal pump were done on CATIA V5. Basic idea of analysis is to study the variation of the pressure head by varying the direction of inlet velocity and by fixing the inlet velocity, outlet pressure and angular velocity of Impeller. The calculations were done in CFD post processor by selecting desired number of time steps and iterations. The pressure head variations can easily be observed in contours generated in Fluent results modular. The standard k-epsilon turbulence model was chosen for turbulence model and by selecting the working fluid as water the simulation was carried out. From the simulation results it was observed that the pressure increases gradually from impeller inlet to outlet, the velocity increases from impeller inlet until it enters the volute casing and it drops to a minimum value at outlet region

***Keywords*— CFD simulation, Catia V5, Inlet Velocity, Impeller Ansys Fluent .**

I. Introduction

Centrifugal pump is a type of a turbo machine in which mechanical energy is converted into pressure energy by means of blade motion and angular momentum change. It is classified as rotor dynamic type of pump in which dynamic pressure is developed which enables the lifting of liquids from lower level to higher level. Since lifting of liquid is due to centrifugal action, it is called as centrifugal pump. Centrifugal pump has high output and high efficiency compared to other types of pumps. To design and develop a centrifugal pump it is very important to predict the final performance with good accuracy and to design prediction of operating characteristics curve is most important. All theoretical methods for prediction of efficiency gives numerical result, but unable to diagnose the root cause for the poor performance. With the development of CFD code it is simple to observe the actual value and easily to get the efficiency value. The prediction pattern and behavioural changes in given physical situation consists of certain values of the relevant variables which govern the processes of interest. With the recent advancement in computing power with powerful graphics and interactive 3D manipulation of models have made the process of developing a CFD model and analysing its results not only reduces the time but it is helpful in terms of labour intensive and ultimately it reduces the cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time.

II. Meshing

In current paper the geometry and the mesh of an eight bladed pump impeller domain were generated with Ansys Workbench. Unstructured meshes with tetrahedral cells are used for the domain of impeller as shown in Figure 1.

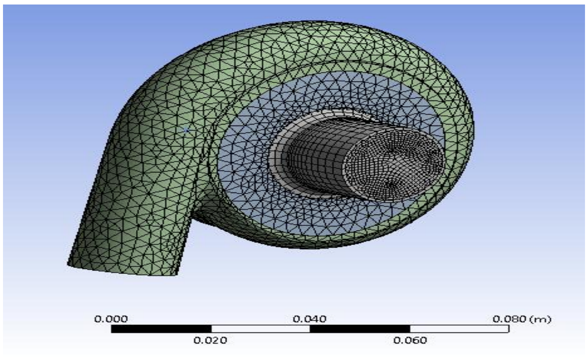

Fig. 1

Mesh Fine mesh

No of Nodes 107728

No of Elements 519137

Simulation of centrifugal pump

The pump assembly was simulated for the pump performance using commercial CFD code and Ansys Fluent after meshing the pump assembly. The boundary conditions were applied. The performance results are obtained at different velocity directions by taking viscous k- epsilon model.

a. Assumptions

The simulation of flow inside the centrifugal pump is done on basis of following basic assumptions:

- Transient flow condition.
- Constant fluid properties.
- Incompressible fluid flow.
- The walls were assumed to be smooth hence any disturbances in flow due to roughness of the surface were neglected.

b. Solution Parameters

Solution parameter is very important in solving any CFD problem. Turbulence numeric is first order. The standard k-ε model is used for turbulence modelling with scalable wall function. The selected working fluid is incompressible water. Number of iteration used for the simulation of centrifugal pump analysis are 8500.

c. Boundary Conditions

Boundary conditions are applied to the pump inlet and outlet with inlet velocity as 0.3 m/sec, outlet pressure as 20000 Pa and angular velocity of impeller as 1800 rpm.

d. Solution Initialization

By providing the initial guess values in the initialization phase of Ansys Fluent the governing equations is solved and the flow field variables are solved by required number of iterations which are desired towards the solution. The hybrid initialization for the velocity and static pressure is used to provide a start point to the solution.

e. Pressure Contours

The properties of internal flow of the centrifugal pump can be analysed by using pressure contours. Study of the pressure contours helps in understanding of energy conversion which took place in different parts of the pump. The variation of Pressure contours with respect to velocity direction is shown in figure below:

- For velocity direction V(1,0,0):

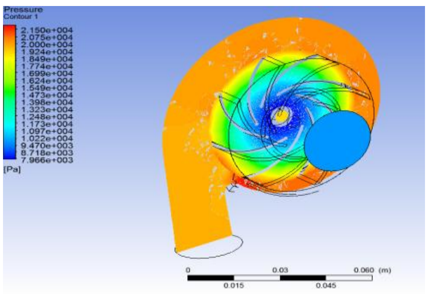

Fig. 2

- For Velocity direction V(1,0,1):

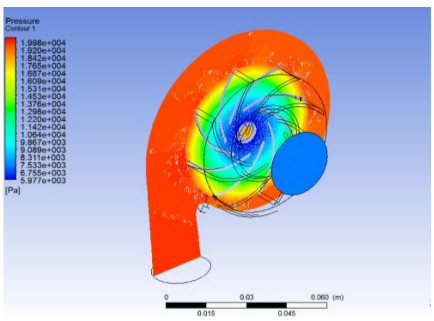

Fig. 3

- For Velocity direction V(1,1,0):

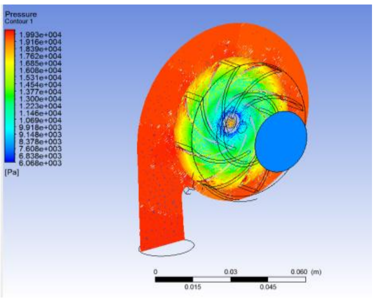

Fig. 4

- For Velocity direction V(1,1,1):

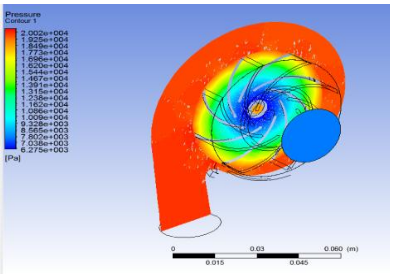

Fig. 5

f. Velocity Vector Contours

Study of the velocity contours gives idea about the kinetic energy and dynamic pressure acting in the different parts. Study of velocity vectors helps in identifying the direction of fluid particles flowing through the different components. The figure below represents the velocity vector contours of designed model.

- For Velocity Direction V(1,0,0):

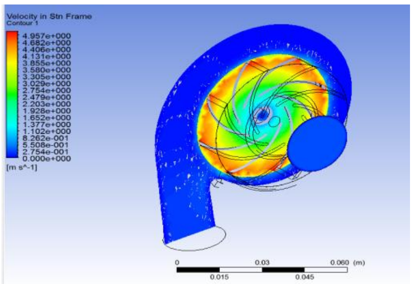

Fig. 6

- For Velocity Direction V(1,0,1):

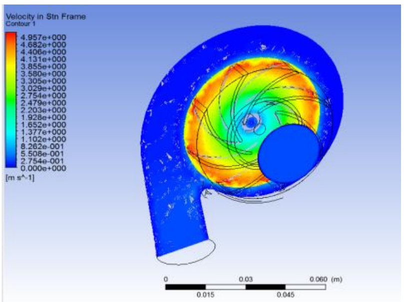

Fig. 7

- For Velocity Direction V(1,1,0):

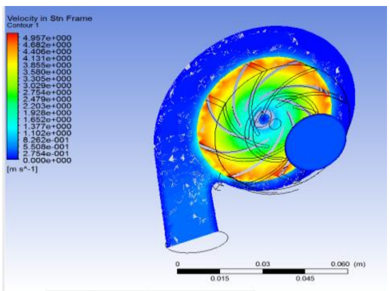

Fig. 8

- For Velocity Direction V(1,1,1):

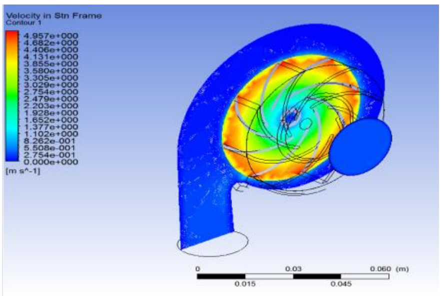

Fig. 9

III. RESULTS

Direction of Inlet Velocity	Inlet Velocity (m/s)	Outlet Velocity (m/s)	Inlet Pressure (Pa)	Outlet Pressure (Pa)	Pressure Head (m)
(1,0,0)	0.3	0.39040	9483.87	20000	1.0723
(1,1,0)	0.3	0.27900	7379.78	19960.9	1.2829
(1,0,1)	0.3	0.27866	7292.20	19960.9	1.2918
(1,1,1)	0.3	0.22792	7602.01	19973.9	1.2602

IV. CONCLUSION

Flow analysis of centrifugal pump is often a challenging task as it requires critical analysis of highly complex flow which is turbulent and three dimensional in nature and having rapidly changing curvature of flow passage. CFD approach has been extensively used in centrifugal pumps as numerical simulation tool for performance prediction at design and off-design conditions, parametric study, cavitation analysis, analysis of interaction effects in different components, prediction of axial thrust, study of pump performance in turbine mode, diffuser pump analysis etc. In our Analysis by fixing the inlet velocity, outlet pressure and angular velocity of Impeller and the variable part being the direction of Inlet Velocity. Our findings through this simulation are as follows:

- Pressure head is high when the inlet velocity co-ordinates are V (1,0,1).
- Pressure head is low when the inlet velocity co-ordinates are V (1,0,0).

By constantly maintaining the Angular velocity of the Impeller we have maximized the Pressure head by varying the direction of Inlet velocity.

REFERENCES

[1] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos, D.P. Margaris, Parametric Study of a Centrifugal Pump Impeller by Varying the OutletBlade Angle, The Open Mechanical Engineering Journal, vol.2, pp.75-83.

[2] 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.

[3] H.Wang, H. Tsukamoto, Fundamental Analysis on Rotor-Stator Interaction in a Diffuser Pump by Vortex Method, Journal of Fluids Engineering123, p. 737.

[4] S. Salvadori, S. Gatta, P. Adami, L. Bertolazzi, Development of a CFD procedure for the axial thrust evaluation in multistage centrifugal pumps.

[5] K. Pagalthivarthi, P. Gupta, V. Tyagi, M. Ravi, CFD Predictions of Dense Slurry Flow in Centrifugal Pump Casings, International Journal of Aerospace and Mechanical Engineering.