ISSN (Online) 2394-2320

International Journal of Engineering Research in Computer Science and Engineering (IJERCSE)
Vol 4, Issue 5, May 2017

Study of Centrifugal Pump Impeller via Ansys CFX Approach

[1] Kapil Rajput

[1] Department of Mechanical Engineering, Galgotias University, Yamuna Expressway Greater Noida, Uttar Pradesh

[1] kapil.rajput@galgotiasuniversity.edu.in

Abstract- Centrifugal pumps are being used most frequently in several fields such as industry, agriculture and domestic use. The most commonly used tool for simulation and analysis is Computational Fluid Dynamics. For the simulation of flow field features within the turbo machinery the 3-D numerical CFD-Tool is used. The flow conditions can be visualized in the centrifugal pump with CFD simulation. ANSYS software package has been used to develop a compressible flow model that is three-dimensional and fully turbulent throughout a complex roller geometry such as the centrifuge pump. It is an automotive and home pump used most commonly. A three-dimensional and fully turbulence-mode Flow through a centrifugal pump impeller. This paper describes the flow simulation in a centrifugal pumps impeller. ANSYS-CFX analyzes the design of centrifugal pump impellers. Centrifugal pump impellers have complex internal flows which can be well anticipated by ANSYS-CFRX.

Keywords- Centrifugal Pump, Impeller, Ansys, Analysis, CFD

INTRODUCTION

Centrifugal pump is a type of turbo engine that uses a centrifugal fluid force to convert mechanical energy into pressure energy. It is a pump type that is classed as rotor-dynamic and develops dynamic pressure that lifts fluids from the lower to the higher. The liquid is called a centrifuge pump because of its centrifugal action. Compared with other kinds of pumps, the centrifugal pump has a high output and efficiency[1]. The efficiency of the flux in the whole pump must be calculated before a stable system for high demand operations is created. This requires a critical analysis of a turbulent and three-dimensional highly complex flow in the pump. CFD[2] simulation enables the flow condition within a centrifugal pump to be visualized and provides precious information on the hydraulic design of the centrifugal pump. Simulation outcome is used for calculating or predicting the centrifugal pump performance in order to replace or reduce pump design experiments. Much work and facilities are saved and help to shorten the design cycle. CFD analysis[3] of the internal flux into a centrifugal pump and its application in pump design processes must therefore achieve significant improvements in the design of the centrifugal pump. Pressure is increased if the velocity is reduced because of the resistance in the system. If resistance is detected, the liquid requires some energy to overcome resistance by means of heat, noise and vibration. As the distance from the pump increases, the energy available in the liquid decreases[4]. A combination of the available velocity and pressure energy at that point is the actual energy for work at any stage in the system.

CONFIGURATION FOR PUMPS

Systemic study on the influence on its performance at various flow rates of the various design aspects of a

ISSN (Online) 2394-2320

International Journal of Engineering Research in Computer Science and Engineering (IJERCSE)
Vol 4, Issue 5, May 2017

centrifugal pump requires numerical predictions and experiments has been tested. In the current analysis, the specifications of the centrifugal pump are shown below. Table 1 shows the specification of proposed model pump.

Table 1. Specification of the Proposed Model Pump

Blade width b	25 mm
Inlet diameter D1	1180 mm
Outlet diameter D2	300 mm
Pump head H	10
Outlet blade angle β^2	20˚
Speed of the impeller N	925 rpm
Flow rate, Q	0.0125 m^3/sec
Specific speed Ns =N√Q/H^4	19.45
The diameter of impeller eye Do= K_0 ^{3}√(Q/N) Where K_0 is the constant 4.5	43.5mm
Hydraulic Efficiency	83%
Number of Blades	6

MESHING

The geometry and mesh is generated using Ansys Workbench in a six bladed pump impeller domain. For the impeller and volute areas also an unstructured mesh is used with tetrahedral cells. The mesh is polished in both the close tongue region of the volute and areas near the border of the blades[5]. Fig 1 shows the meshing structure. Structured hexaedral cells are created around the blades to provide more details of the boundary layer. The mesh above the area of the tongue appears in Fig.2. The impeller domain has a minimum of 3575036 elements[6].

Figure 1. Meshing Structure

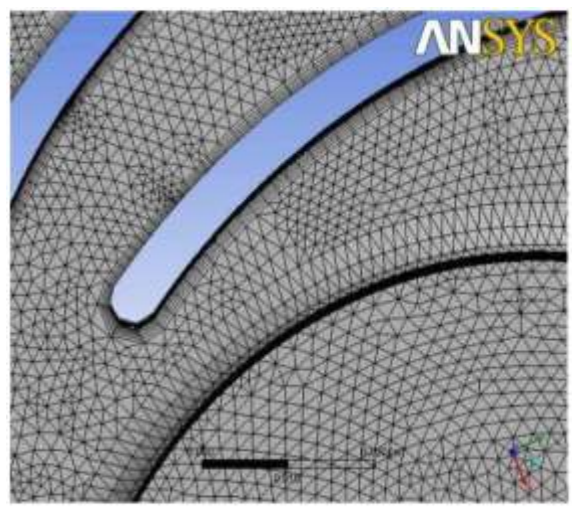


Figure. 2 Meshed Area

With Ansys Workbench[7], the geometry and mesh of a domain of six pump impellers is created. For the impeller and volute zones as shown on fig.1, an unstructured mesh with tetrahedral cells is also used. The mesh is polished both in the region near the tongue in the volute and in places near the border of the blades. Structured hexahedral cells are created around the blades to provide more detailed boundary layers[8]. The mesh near the tongue area is illustrated in Fig.3. The impeller domain has a total of 3575036 elements. Table 2 shows the statistics on the mesh.

Table 2. Mesh statistics

Number of nodes	806744
Number of tetrahedral	3417824
Number of prisms	251367
Number of elements	3575301
Number of pyramids	918

OPTIMIAL CONDITION

The pump impeller domain centrifugal is regarded as the reference rotational frame at 925 rpm rotational velocity. Water at 27°C is the working fluid in the pump. The 5% k-µ turbulence rate model is considered. Boundary conditions include static pressure and mass outlet intake of 14.5 kg / s. The Ansys-CFX solver is used to solve 3 in-compressible N-S equations.

RESULTS

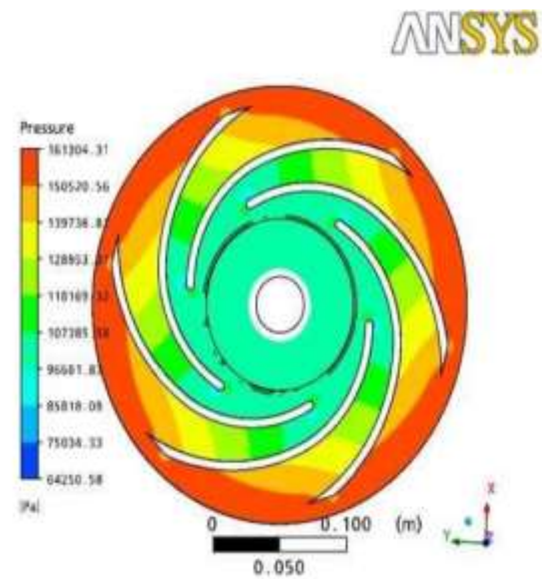
The designed mass flow rate of 14.5 kg / m3 centrifugal pump impeller with no volute housing is resolved. Table No.3 shows the performance results. The total efficiency obtained from the CFD analysis is 9.6517 m / WG and 90.9029 percent. The table 3 shows the performance results for the pump.

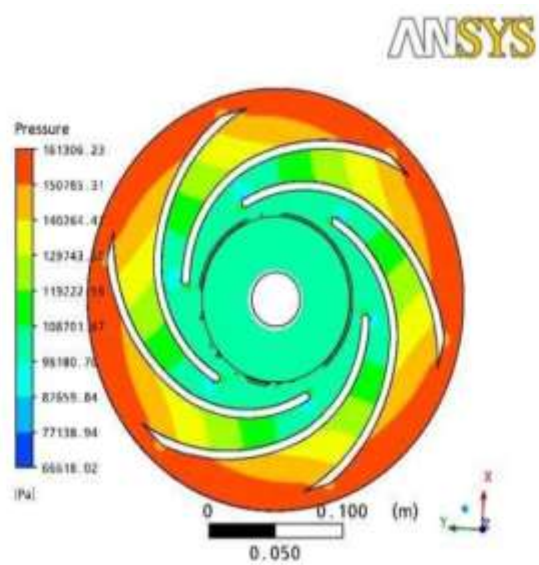
Table 3. The Performance of Result of the Pump

Sl.No	Descriptions	parameters
1.	Diameter of causing	0.280 m
2.	Rotational speed	942 rpm
3.	Volume of flow rate	0.0125 m^3/s
4.	Head (in)	9.25 m
5.	Head (out)	9.45 m
6.	Flow Coefficient	0.0931
7.	Head Coefficient	0.134
8.	Shaft power	14209.01 w
9.	Power Coefficient	0.0145
10.	Static Efficiency %	64.56
11.	Total Efficiency %	95.05

Pressure variation on the proposed model- The spatial properties of the internal flow field are measured in comparison to the performance of the centrifuge pump. Pressure contours at 10, 25, 50, 75, 90 and 100 percent are shown in Fig.4 at various locations. The fluid head of the spinning pump rotator, the friction contours indicate a constant pressure change from the lead edge to the trailing edge of the roller. The total pressure on the blade's side is more than the suction side[9]. It is observed. From the tip to the bottom of the blade, the difference in pressure between the contact side and the suction side decreases. On the front of the blades on the suction side is the minimum value of the static pressure inside the impeller. At the center of the impeller, low total pressures are found[10]. The total strain is increasing as the blade edge becomes highly dynamic, with the duration increasing. At the leading edge on the suction side of the blade is low total pressure and high velocity due to the vane size. The total pressure loss at the trailing edge of the blade is observed at every point due to the wake formation at the trailing edge of the blade.

International Journal of Engineering Research in Computer Science and Engineering (IJERCSE)
Vol 4, Issue 5, May 2017





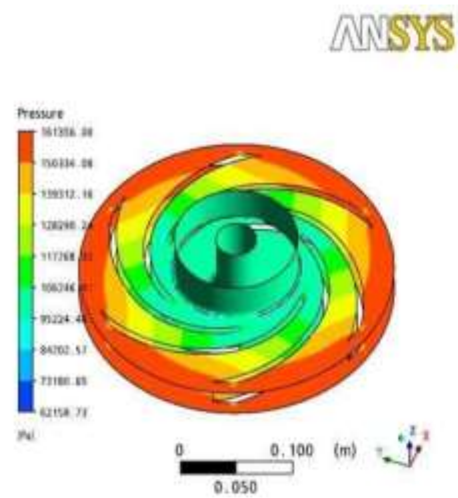


Figure 3. Pressure Counter of the Pump

CONCLUSIONS

A pump impeller is designed and resolved by means of computerized fluid dynamics, pump flow patterns, performance results, the circumferential area averaged pressure, plot loading for a blade at 50%, weak changes in average total mass pressure and static pressure, wise flux variation of the average area absolute speed and variation of m m A centrifugal pump impeller is modeled and solved.

REFERENCES

[1] X. Chen and Y. Liu, *Finite element modeling and simulation with ANSYS workbench*. 2014.

[2] A. . Fallis, "Turbulence Modelling for CFD," *J. Chem. Inf. Model.*, 2013.

[3] S. Jamshed, "Introduction to CFD," in *Using HPC for Computational Fluid Dynamics*, 2015.

[4] L. Davidson, "Fluid mechanics, turbulent flow and turbulence modeling," *CFD course*, 2012.

[5] A. Fluent, "Ansys Fluent Theory Guide," *ANSYS Inc., USA*, 2013.

[6] I. ANSYS, "ANSYS® Academic Research," *ANSYS CFX-Solver Model. Guid.*, 2013.

[7] S. Meshing, "Introduction to ANSYS Meshing," *Workbench*, 2010.

[8] B. Blocken, T. Stathopoulos, and J. Carmeliet, "CFD simulation of the atmospheric boundary layer: wall function problems," *Atmos. Environ.*, 2007.

[9] S. R. Shah, S. V. Jain, R. N. Patel, and V. J. Lakhera, "CFD for centrifugal pumps: A review of the state-of-the-art," in *Procedia Engineering*, 2013.

[10] V. S. Lobanoff and R. R. Ross, *Centrifugal Pumps: Design and Application*. 2013.

11. V.M.Prabhakaran , Prof.S.Balamurugan , S.Charanyaa, "A Strategy for Secured Uploading of Encrypted Microdata in Cloud Environments", International Advanced Research Journal in Science, Engineering and Technology Vol. 1, Issue 3, November 2014

[12] R Santhya, S Balamurugan, "A Survey on Privacy Preserving Data Publishing of Numerical Sensitive Data", International Journal of Innovative Research in Computer and Communication Engineering , Vol. 2, Issue 10, October 2014

[13] Balamurugan Shanmugam, Dr.Visalakshi Palaniswami, Santhya. R, Venkatesh. R.S., "Strategies for Privacy Preserving Publishing of Functionally Dependent Sensitive Data: A State-of-the art Survey. Aust. J. Basic & Appl. Sci., 8(15): 353-365, 2014

[14] Gagandeep Singh Narula, Dr. Vishal Jain, Dr. S. V. A. V. Prasad, "Use of Ontology to Secure the Cloud: A Case Study", International Journal of Innovative Research and Advanced Studies (IJIRAS), Vol. 3 No. 8, July 2016, page no. 148 to 151 having ISSN No. 2394-4404.