International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 7, July - 2014

Design And Flow Through CFD Analysis Of Enclosed Impeller

Ashish J. Patel[1], Bhaumik B. Patel[2]
[1] M.E (Machine Design) Student, Mechanical Engineering,
Kalol Institute of Technology and Research Centre, Kalol, Gujarat, India
[2] Assistant Professor, Mechanical Engineering Department,
Kalol Institute of Technology and Research
Centre, Kalol, Gujarat, India,
Gujarat Technological University

Abstract -- The Impeller of Centrifugal Pump is one of most Critical Component to be designed. The Flow through Centrifugal Pump impeller is three dimensional and fully turbulence model. The present work describes the design and flow through CFD analysis of enclosed impeller of the centrifugal Pump. The model of impeller was generated using SOLIDWORKS and analyzed in ANSYS (Workbench) Fluid Flow (CFX). The numerical solution of the discredited three dimensional, incompressible Navier-Stokes equations over an unstructured grid is accomplished with an ANSYS-CFX. The design of enclosed impeller is done by using various input data, the value of head is determined by theoretical as well as CFD value. The CFD analysis of Enclosed impeller has been carried out at various inlet and outlet blade angles of the impeller and number of blades of impeller to investigate the changes in head as well as efficiencies.

Keywords – Impeller Design, Solid Works, CFD, Pressure Distribution, Efficiency

1. INTRODUCTION

A pump is a machinery or device for raising, compressing or transferring fluid. A fluid can be gasses or any liquid. Pumps are one of the most often sold and used mechanical devices and can be found in almost every industry. Pumps alone do not create pressure; they only displace fluid, causing a flow. Adding resistance to flow causes pressure. Pumps fall into five major groups: direct lift, velocity, buoyancy and gravity pumps.

A centrifugal pump is a kinetic device. Liquid entering the pump receives kinetic energy from the rotating impeller. The centrifugal action of the impeller accelerates the liquid to a high velocity, transferring mechanical (rotational) energy to the liquid. That kinetic energy is available to the fluid to accomplish work. The centrifugal pumps act as a reversed of an inward radial flow reaction turbine. Centrifugal pumps consist of a set of rotating vanes, enclosed within a housing or casing, used to impart energy to a fluid through centrifugal force.

2. OBJECTIVE

To Perform Design and Flow through CFD analysis of the Enclosed impeller in Ansys software with modification in inlet and outlet blade angles of the impeller and number of blades of the impeller to investigate the changes in head as well as efficiencies.

3. LITERATURE REVIEW

E.C. Bcharoudis et. al. [1] have contributed to reveal the flow mechanisms inside centrifugal impellers and studied performance by varying outlet blade angle. They observed a gain in head more than 7 % with increase in outlet blade angle from 20 degree to 45 degree.

K. M. Pandey et. al. [2] have performed two-dimensional steady numerical analysis for centrifugal pumps with impeller blades 7, 8 and 9 using Ansys Fluent 6.3 software for inlet diameter 80 mm and outlet diameter 168 mm at 2500 rpm rotational speed also to investigate the changes in head as well as efficiencies with the increase of blade number.

S.Rajendran et. al. [3] have performed the simulation of the flow in the impeller of a centrifugal pump. The flow pattern, pressure distribution in the blade passage, blade loading and pressure plots are discussed.

4. Design of Enclosed Impeller

For design calculation, the design Parameters of CROMPTON SRAM-400 Pump are taken as follows:

Table 4.1 Design Data for Pump

Sr. No	Parameters	Value
1	Flow rate	0.5 m^3/s
2	Head	30 m
3	Pump Speed	735 rpm
4	Gravitational Acceleration	9.81 m/s^2
5	Density of Sewage Water	1050 kg/m^3

The design steps are as follows: [4]

1. Calculation of Shape Number (N_{sh}):

$$N_{sh} = \frac{10^3 * n * \sqrt{Q}}{(g*h)^{\frac{3}{4}}}$$

2. Calculation of Power (P):

- Power input to the pump, P_c is given by,

$$P_C = \frac{\rho * g * H * Q}{\eta_{ov}}$$

- Calculation of power to be supplied by the Motor P_{Cm},

$$P_{cm} = (M_{olf}) * P_C$$

Where (M_{olf}) is overloading factor.

3. Calculation of Shaft Diameter (d_{sh}):

$$d_{sh} = \sqrt[3]{\frac{16 * P_{cm}}{\omega * \pi * \tau_{tor}}}$$

4. Calculation of Hub Diameter (d_h):

$$d_h = 1.4 * d_{sh}$$

5. Calculation of Eye Diameter (d_e):

$$d_e = 4.5 * \sqrt[3]{\frac{Q}{n}}$$

6. Impeller inlet diameter (D_i):

$$D_i = 1.05 \text{ to } 1.02\, d_e$$

7. Calculation of inlet vane angle (β_1):

$$\beta_1 = \tan^{-1}\left(\frac{C_{m1}}{u_1}\right)$$

8. Tangential velocity at inlet (U_1):

$$U_1 = \frac{\pi * D_i * n}{60}$$

9. Calculation of number of Vanes (Z):

$$Z = \frac{\left(\frac{6.5*(D_2+D_1)*\sin(\beta_1+\beta_2)}{(D_2-D_1)}\right)}{2}$$

10. Calculation of Suction Pipe Diameter (D_S):

$C_{m1} = C_0$,

Where C_{m1} = Flow velocity at inlet

But, $C_0 = 0.06 \text{ to } 0.08 * \sqrt[3]{Qn^2}$

11. Inlet width at the impeller (B_1):

- The flow area just inside the vane passage at the inlet, is

$$A_1 = \frac{\Phi_1 * Q'}{C_{m1}}$$

- Vane contraction factor at the inlet,

$$\Phi_1 = \frac{t_1}{t_1 - S_{u1}}$$

Where, t_1 = Pitch of vanes

S_{u1} = Peripheral vane thickness at the inlet

$$t_1 = \frac{\pi * D_i}{Z}$$

- Peripheral Vane thickness at inlet,

$$S_{u1} = \frac{S_1}{\sin\beta_1}$$

Where, S_1 = Vane thickness at inlet 5 to 8 mm.

$$B_1 = \frac{A_1}{\pi * D_i}$$

12. Vane angle at the outlet (β_2):

$$\beta_2 = 35 - \frac{N_{sh}}{8}$$

13. Calculation of outlet diameter (D_O):

$$U_2 = \frac{C_{m2}}{2\tan\beta_2} + \left(\left(\frac{C_{m2}}{2\tan\beta_2}\right)^2 + gH_{b1\infty}\right)^{1/2}$$

- Hydraulic efficiency η_{hy},

$$\eta_{hy} = 1 - \frac{0.42}{(\log d_e - 0.172)^2}$$

- Theoretical Head H_{b1},

$$H_{b1} = \frac{H}{\eta_{hy}}$$

- Flow velocity at outlet, $C_{m2} = 0.8 \text{ to } 0.9 C_{m1}$

Substituting the value of U_2, $D_o = \frac{U_2 * 60}{\pi * n}$. Similar steps can be applied for obtaining the outlet width of blade (B_2) as for inlet width (B_1).

14. Relative velocity at inlet and outlet (V_r):

$$V_{r1,2} = \frac{C_{m1,2}}{\sin\beta_{1,2}}$$

The Calculated Parameters are as below:

Sr.	Parameters	Value
1	Impeller inlet diameter	402 mm
2	Impeller outlet diameter	630 mm
3	Impeller inlet width	120 mm
4	Impeller outlet width	70 mm
5	Blade thickness	15 mm
6	Inlet vane angle	14^0
7	Outlet vane angle	20^0
8	Number of vanes	4

5. CFD ANALYSIS OF ENCLOSED IMPELLER

Computational Fluid Dynamic is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problem that involve fluid flows.

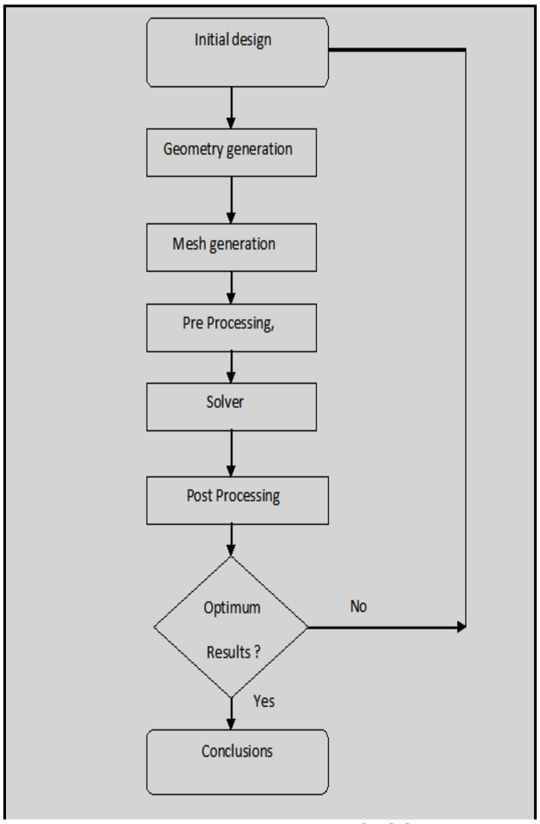


Figure 5.1 CFD methodology

6. BOUNDARY CONDITIONS

Centrifugal pump impeller domain is considered as rotating frame of reference with a rotational speed of 735 rpm. The working fluid through the pump is water at 27 ^{0}C. k-ε turbulence model with turbulence intensity of 5% is considered. Inlet and outlet Pressure and mass flow rate of 0.5 m^3/s are given as boundary conditions. Three dimensional incompressible N-S equations are solved with Ansys-CFX Solver.

7. PROCEDURE OF CFD ANALYSIS

Step1. 3D Model of Impeller is generated in SOLIDWORKS 2009 as per above given Drawing.

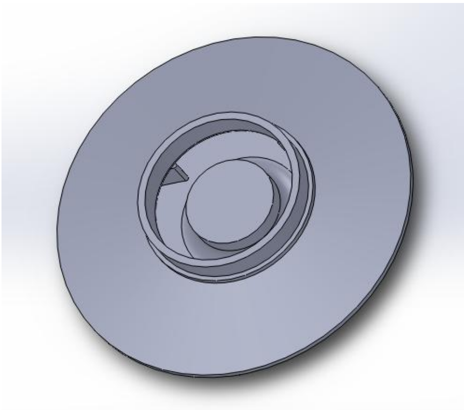


Figure 7.1 3D Model of Impeller

Step2. Our CFD Analysis method is Cavity Patten so we have to create Cavity model of above impeller.

Step3. Save above Cavity model in *. IGES Format for Importing into ANSYS Workbench Mesh Module for Meshing.

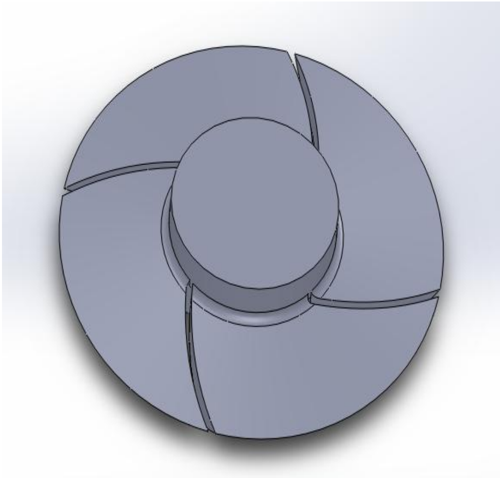


Figure 7.2 Cavity Model of Impeller

Step4. Import above Cavity model in ANSYS Workbench Mesh Module.

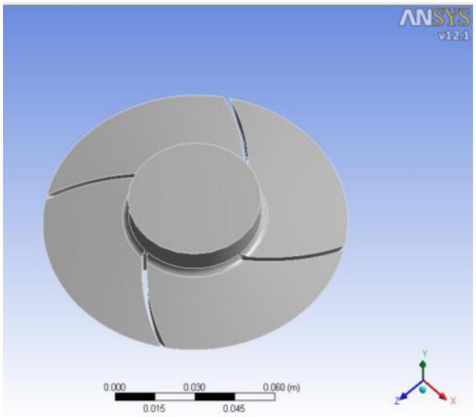
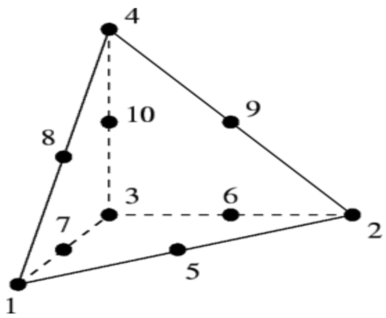


Figure 7.3 Cavity model of impeller in ANSYS Workbench

Step5. Meshing of Impeller
Meshing Type: 3D
Type of Element: Tetrahedral



No. of Nodes: 46403
No. of Elements: 241146

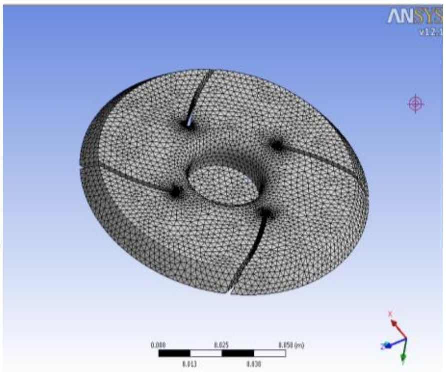


Figure7.4 Meshed Model of Impeller Cavity

Step6. Save above model in *.CMDB Format for importing into ANSYS CFX Pre.

Step7. Import above .CMDB File in ANSYS CFX Pre.

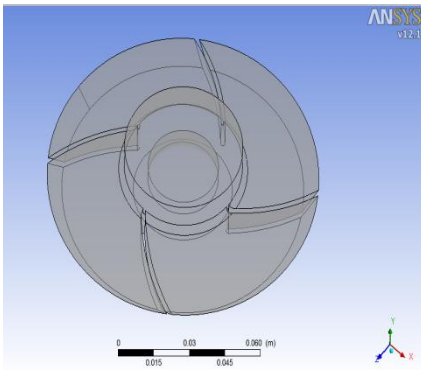


Figure 7.5 Impeller Cavity in ANSYS CFX Pre

Step8. Define Water + Particle Domain.
Domain Type: Fluid
Domain Fluid: Water + 5 mm Particle
Domain motion: Rotating
Domain RPM: 735 rpm
Rotating about: Y Axis

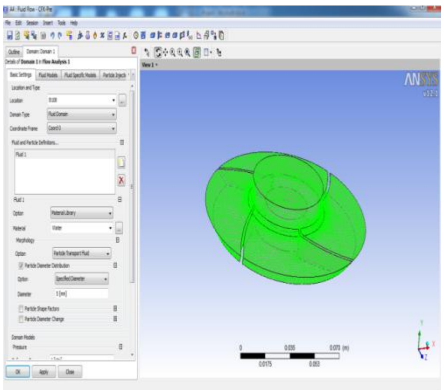


Figure 7.6 Water + Particle Domain

Step9. Define Heat Transfer and Turbulence model.

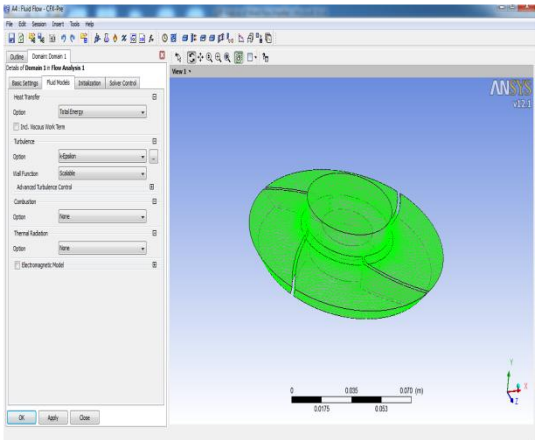


Figure 7.7 Heat Transfer and Turbulence Model

Heat Transfer Model: Total Energy
Turbulence Model: k- epsilon
Where k is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. It has dimensions of (L2 T-2); for example, m^2/s^2.
ε is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate), as well as dimensions of k per unit time (L2 T-3) (e.g., m^2/s^3).
The k-ε model introduces two new variables into the system of equations. The continuity equation is then:

$$\frac{\partial \rho}{\partial t} + \nabla \bullet (\rho U) = 0$$

and the momentum equation becomes

$$\frac{\partial \rho U}{\partial t} + \nabla \bullet (\rho U \otimes U) - \nabla \bullet (\mu_{eff} \nabla U) = \nabla P' + \nabla \bullet \left(\mu_{eff} \nabla U\right)T + B$$

Step10. Define Hub as a Rotating Wall
Wall Roughness: Smooth Wall
Heat Transfer: Adiabatic

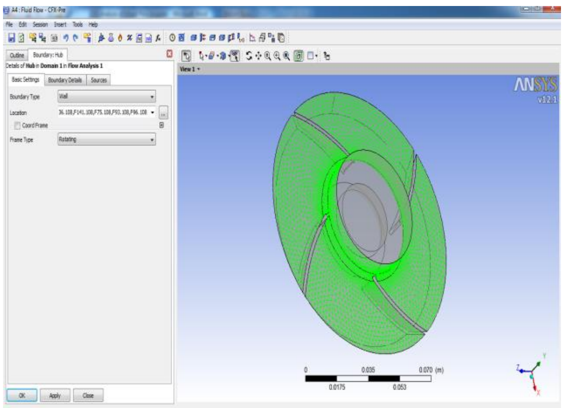


Figure 7.8 Hub as a Rotating Wall
Step11. Define Shroud as a Rotating Wall.

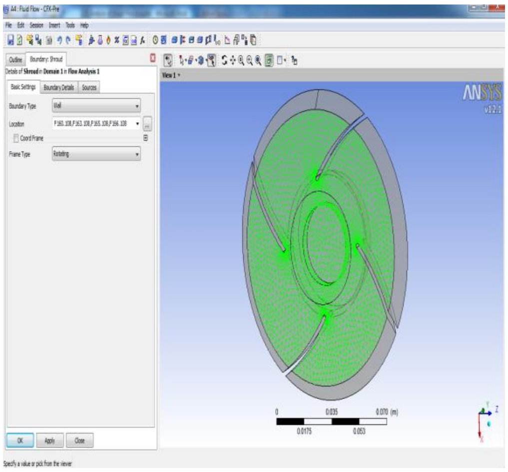


Figure 7.9 Shroud as a Rotating Wall

Step12. Define inlet for Impeller
Define inlet mass Flow Rate: 0.5 m^3/s
Static Frame to Total Temperature: 300 K

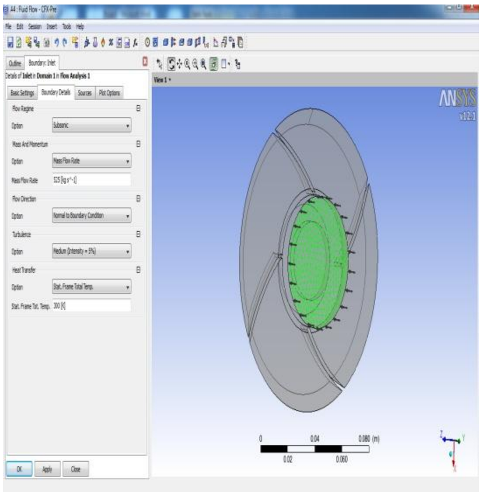


Figure 7.10 Inlet for Impeller

Step13. Define Outlet for Impeller
Define Outlet mass Flow Rate: 0.5 m^3/s

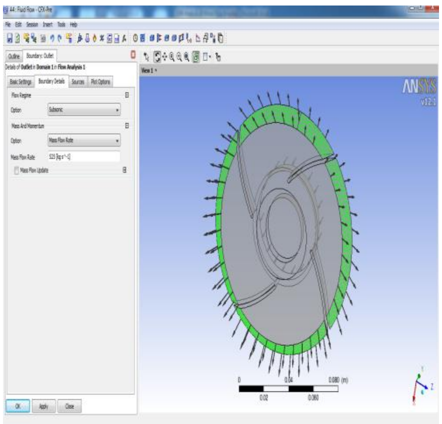


Figure 7.11 Outlet for Impeller

Step14. Define Solver Control Criteria.

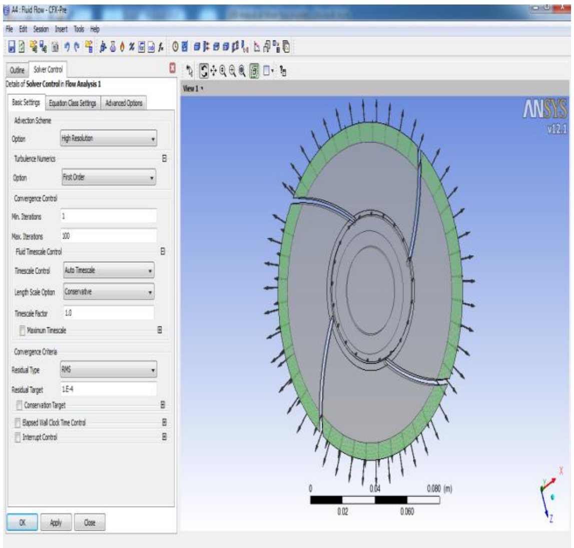


Figure 7.12 Solver Control Criteria

Number of Outer loop iteration: 100
Convergence Criteria:
Residual Target: 1e-4
Step15. Run the Analysis

7.2 Results of Analysis:

Inlet Angle: 14^0
Outlet Angle: 20^0

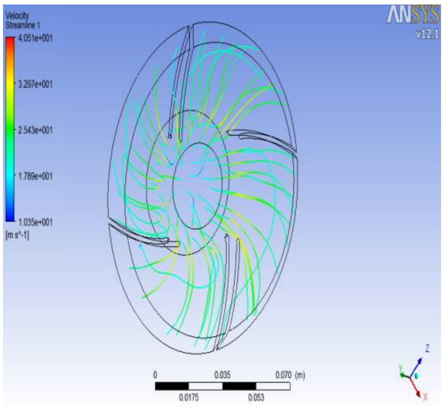


Figure 7.13 Velocity Contour

Figure 7.13 shows minimum velocity is $1.035*10^1$ m/s and maximum velocity is $4.051*10^1$ m/s.

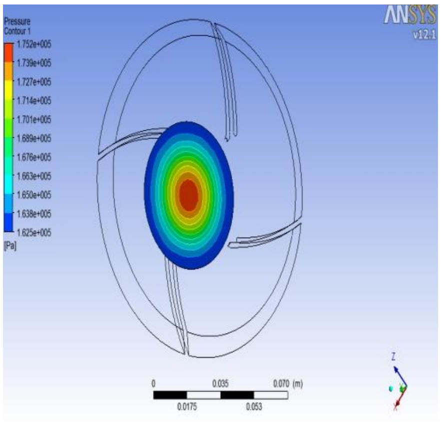


Figure 7.14 Inlet Pressure Contour

Figure shows pressure contours at inlet, it helps one in identifying the maximum and minimum pressure at the inlet when the fluid comes in contact with the blade. Here minimum inlet pressure is $1.625*10^5$ Pascal and maximum inlet pressure is $1.752*10^5$ Pascal.

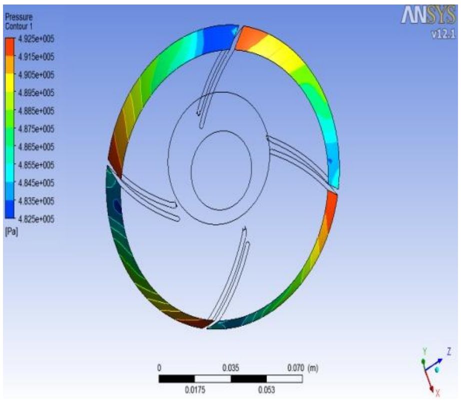


Figure 7.15 Outlet Pressure Contour

Figure shows that pressure at outlet of impeller, before entering the casing. Here minimum outlet pressure is $4.825*10^5$ Pascal and maximum outlet pressure is $4.925*10^5$ Pascal.

$$\text{Head} = \frac{\text{Outlet Pressure} - \text{Inlet Pressure}}{\rho * g}$$

$$= \frac{4.925e5 - 1.752e5}{1050 * 9.81}$$

Head = 30.804 m

7.3 Modification in Inlet and Outlet Blade Angles of Impeller

Here we took various inlet and outlet blade angles to show pressure distribution at inlet and outlet of impeller before entering the casing. Among them the best result was taken and discussed below.

Inlet Angle: 14^0
Outlet Angle: 18^0

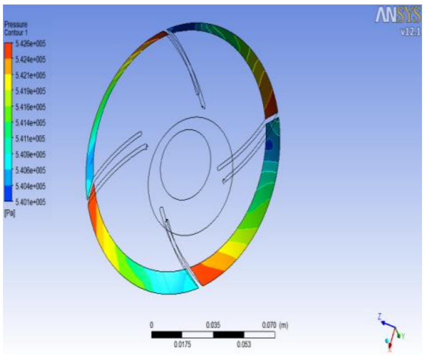


Figure 7.16 Outlet Pressure Contour

Here minimum outlet pressure is $5.401*10^5$ Pascal and maximum outlet pressure is $5.426*10^5$Pascal. Figure 4.29 shows inlet pressure contour at inlet blade angle 14^0 and outlet blade angle 18^0 at which fluid leave the impeller and pump get maximum efficiency (approximately 96%).

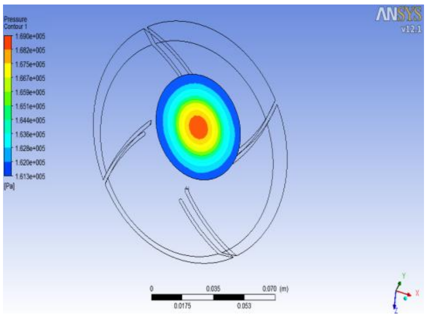


Figure 7.17 Inlet Pressure Contour

Here minimum inlet pressure is $1.613*10^5$ Pascal and maximum inlet pressure is $1.690*10^5$Pascal. Figure 4.30 shows inlet pressure contour at inlet blade angle 14^0 and outlet blade angle 18^0 which indicate maximum pressure at which fluid comes in contact with the impeller and the head is maximum(approximately 36 m).

$$\text{Head} = \frac{\text{Outlet Pressure} - \text{Inlet Pressure}}{\rho * g}$$

= (5.426 e5 - 1.69 e5)/(1050 * 9.81)
= 36.27 m

Table 7.1 Generated Head at Various Blade Angles

Sr. No	Inlet Angle	Outlet Angle	Head (m)	η (%)
1	16^0	22^0	33.52	90.86
2	16^0	20^0	34.59	93.76
3	16^0	18^0	35.27	95.60
4	14^0	22^0	28.34	76.82
5	14^0	20^0	30.80	81.00
6	14^0	18^0	36.27	96.68
7	12^0	22^0	26.15	70.88
8	12^0	20^0	28.96	78.50
9	12^0	18^0	36.12	95.94

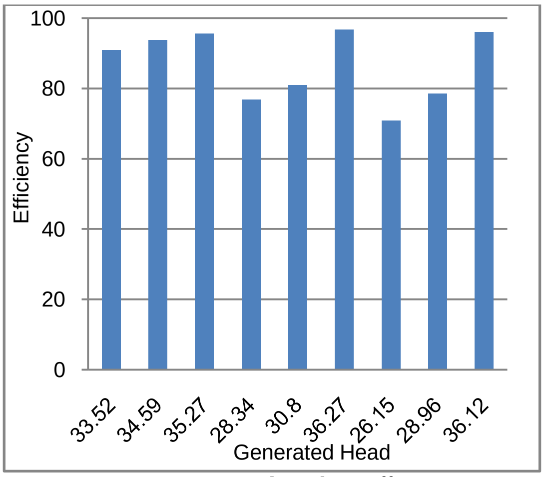


Figure7.18 Generated Head Vs. Efficiency

7.4 Modification in Number of Blades of Impeller

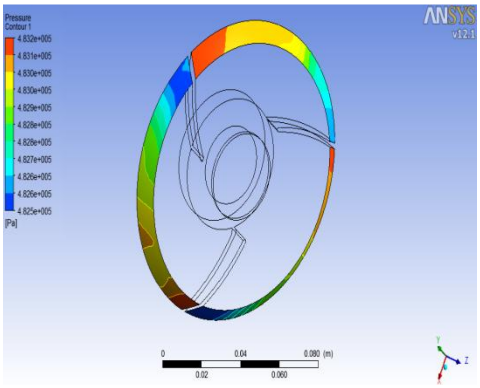


Figure 7.19 Outet Pressure Contour at Number of Blades 3

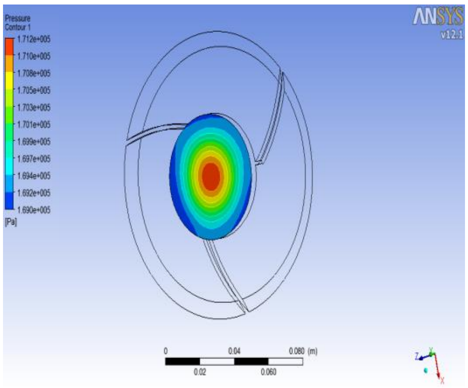


Figure 7.20 Inlet Pressure Contour at Number of Blades 3

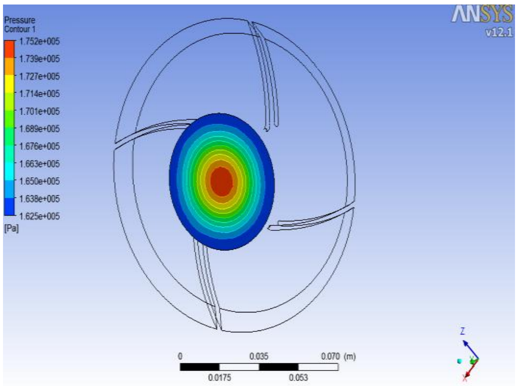


Figure 7.21 Inlet Pressure Contour at Number of Blades 4

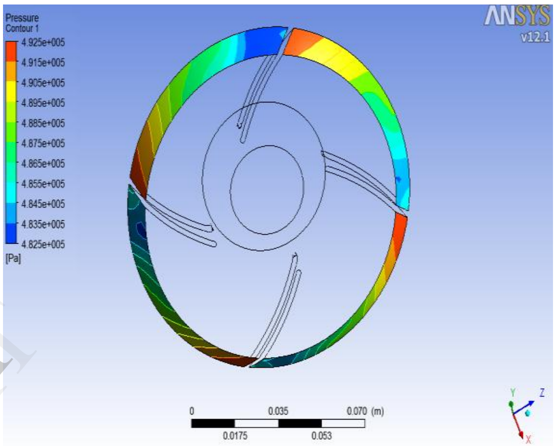


Figure 7.22 Outlet pressure Contour at Number of Blades 4

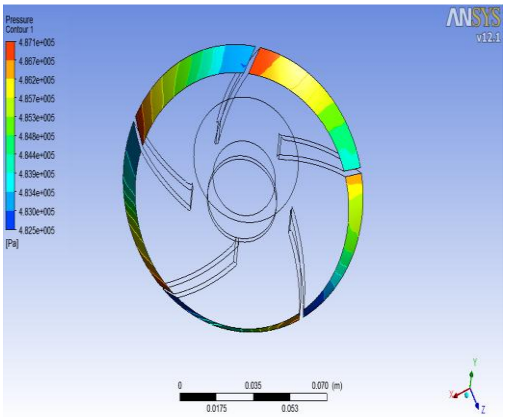


Figure 7.23 Outlet pressure Contour at Number of Blades 5

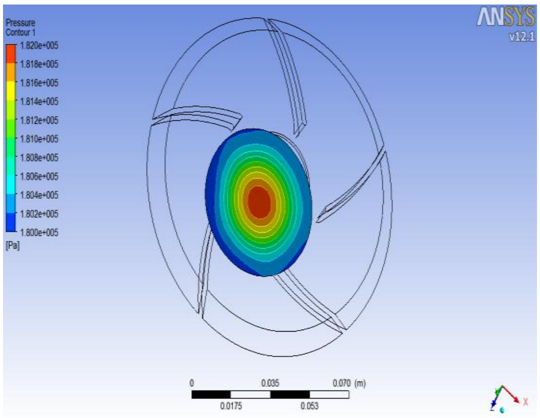


Figure 7.24 Inlet pressure Contour at Number of Blades 5

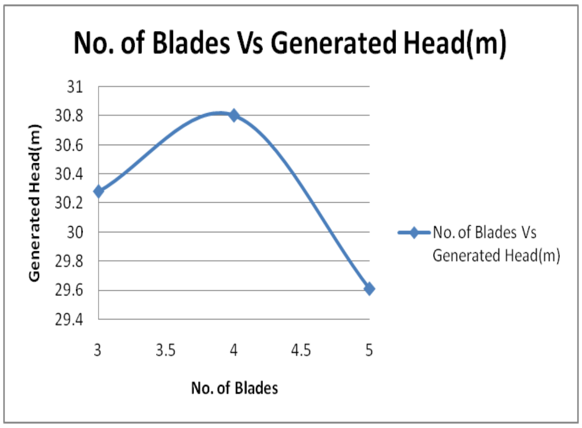


Figure7.25 Number of Blades Vs. Head

CONCLUSION

The CFD predicted value of the head at the designed flow rate is approximately H=30.804 m. There is 8.04% of difference between the theoretical head and the predicted numerical head.

From the CFD analysis of centrifugal pump impeller, the maximum generated head is 36.27m at inlet blade angle 14^0 and outlet blade angle 18^0 respectively and the pump efficiency is approximately 96%.

With modification in number of blades of impeller, it is conclude that head is decreases at number of blades 3 and 5. So number of 4 blade impeller is suitable for this pump.

REFERENCES

[1] 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, 2008, 2, 75-83.

[2] K. M. Pandey, A. P. Singh and Sujoy Chakraborty, “Numerical studies in effect of Blade number variations on performance of Centrifugal Pumps at 2500 rpm”, Journal of Environmental Research and Development, Vol.6 No. 3A, Jan-March 2012.

[3] S.Rajendran and Dr.k.Purushothaman, “Analysis of a centrifugal pump impeller using ANSYS-CFX”, International Journal of Engineering Research & Technology (IJERT) Vol. 1 Issue 3, May – 2012, ISSN: 2278-0181.

[4] Prof. S. Kumaraswamy. “Databook for design of Centrifugal Pumps”. Center for Industrial Consultancy and Sponsored Research, IIT Madras.

[5] Sujoy Chakraborty, Kishan Chaudhary, Prasenjit Dutta, “Performance prediction of Centrifugal pumps with variations of blade number”, Journal of science and Industrial research, vol.72, June 2013, pp. 373-378.

[6] Weidong zhou, Zhimei Zhao, T. S. Lee, and S. H. Winoto, “Investigation of flow through Centrifugal Pump Impellers using Computational Fluid Dynamics”, International Journal of Rotating Machinery, 9(1): 49-61, 2003.

[7] C. P. Kothandaraman, R. Rudramoorthy, “Fluid Mechanics And Machinery (Second Edition)”, Handbook, New Age International (P) Limited, Publishers.

[8]G.K. Sahu., Pumps, New Age International Publishers, First edition, 2000.

[9] John D. Anderson Jr., Joris Degroote, G´erard Degrez, Erik Dick, “Computational Fluid Dynamics an Introduction”, Handbook, Springer.