Improving Efficiency of Submersible Pump Impeller of Mixed Flow Type by Design Modification through CFD Analysis

Naveen Nagalinga Lohar[1], Prof. S A Janawade[2].

[1]Department of Mechanical Engineering (M.Tech in Design Engineering)
KLE Dr. M S Sheshgiri College of Engineering and Technology Udyambag, Belagavi, Karnataka, India 590008
[2] Department of Mechanical Engineering (Professor)
KLE Dr. M S Sheshgiri College of Engineering and Technology Udyambag, Belagavi, Karnataka, India 590008

---***---

Abstract - Subject of this dissertation work is based on mixed flow pump. Mixed flow pumps have wide range of applications in fields like agricultural irrigation, flood control etc, and also in domestic utilities. But losses occurred inside pump reduces efficiency of mixed flow pump and they also consume large electricity. So by increasing efficiency of pump for the same energy consumption, energy can be saved. So to improve performance of mixed flow pump work is carried out at design level and parameters affecting design and performance of pump are identified. Then to improve efficiency of pump, first analytical designing is done for new improved efficiency pump impeller. These analytical solutions are then used to model the new improved impeller. In ANSYS Blade Modeller's Vista CPD under ANSYS Workbench 1D model is created. Solid model of impeller is done using ANSYS BladeGen. CFD analysis is carried out in ANSYS CFX. The results of CFD analysis of improved efficiency impeller is compared with analytical solutions, for justification of results which shows close match between both results. Then the results of improved efficiency impeller are compared with existing pump impeller and it is found that by changing inlet and exit angles of impeller blade, reducing number of blades and backward curved vanes improve pump performance significantly.

Key Words: Mixed flow pump impeller. Efficiency. Pump parameters, CFD analysis, ANSYS CFX.

1. INTRODUCTION

Pump is a device that uses mechanical energy to move the fluid. Pumps play a very important part in many domestic and industrial applications. Different types of pumps are used in many applications. Pumps are classified mainly into 3 categories based the method they use for lifting fluid and they are gravity, direct lift and displacement pumps. Depending on the configuration of impeller and specific speed pumps are classified as radial, mixed and axial flow pump. Pumps work either rotary or reciprocating mechanism.

1.1. BROAD VIEW OF PUMPS

Broadly pumps are classified into two types according to their operating method and they are dynamic pumps and positive displacement pumps. A positive displacement pump makes liquid move by trapping a fixed quantity of liquid and driving that volume of liquid into exit pipe. They are also called steady flow pumps.

In dynamic pumps by increasing the velocity of flow kinetic energy is transferred to the liquid, so these are called velocity type pumps. Dynamic pumps are classified into 3 types and they are axial flow pump, radial flow pumps and mixed flow pumps. Radial flow pumps are centrifugal pumps and in these pumps fluid leaves the impeller in radial direction and enters the pump at the centre. In axial flow pumps fluid particles do not change their path because at the entry and exit change in radius is very small.

Mixed flow pumps (MF pumps) are used in large scale among different types of pumps available. In these pumps head is partly developed partly by centrifugal action and partly by bane lift. In these pumps liquid enters axially and leaves in radial way. Impeller is an important part of a submersible pump and pump performance depends on geometry of impeller. Pump performance can be improved by changing the parameters of the pump. Produced liquids, after being subjected to great centrifugal forces caused by the high rotational speed of the impeller, lose their kinetic energy in the diffuser where a conversion of kinetic to pressure energy takes place. This is the mechanism of radial and mixed flow pumps.

2. PROBLEM IDENTIFICATION AND OBJECTIVES

Centrifugal pumps are widely used but their performance is generally below the expected level. So it is important to improve performance. Performance can be improved by changing the vane profile and by changing geometric parameters. Following Table 2.1 gives the existing impeller data

Table -2.1: Existing impeller data

Quantity(symbol)	Value	Unit
Head(H)	46	Meter
Discharge(Q)	0.0078	m³/sec
Rotational speed(n)	2880	Rpm
No of stages(m)	6	No

2.1. Objectives
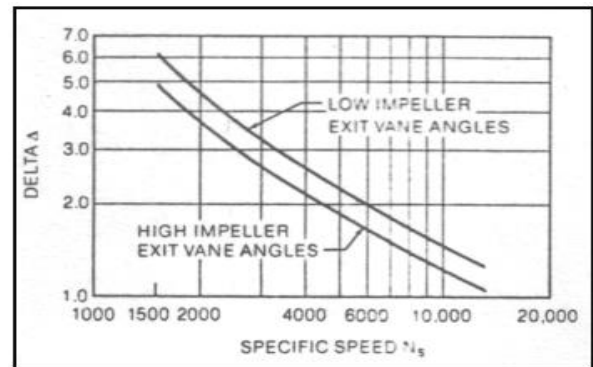
- Designing mixed flow impeller using ANSYS B lade design module.
- To use CFD software in designing impeller.
- To make comparison between analytical and CFD solutions.
- To check the effect of design parameters on head and efficiency by CFD analysis

3. DESIGN OF IMPELLER

3.1 Design of Impeller Layout Profile

Specific speed selection has to be done in U.S. customary units for selection of new impeller by using existing impeller parameters.

$$N = \frac{n\sqrt{Q}}{H^{0.75}}$$

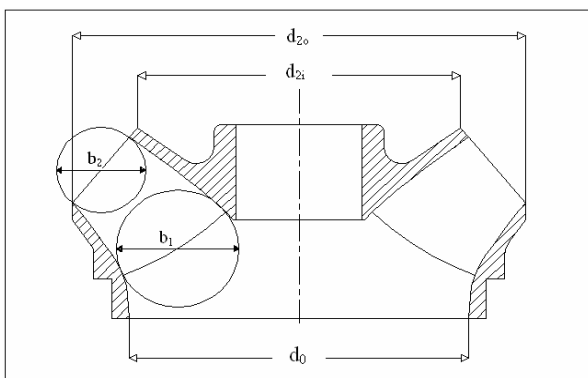From the above equation value of N comes out to be 2527.72rpm.

Figure 3.1. Selected impeller layout profile from Layne Bowler Library

For above value of $N_{(u\,s)}$, we get Δ_{high} = 3.2 and Δ_{low}= 3.8 as shown in figure 3.2 of Cordier diagram.

Figure 3.2. Cordier Diagram for Δ value selection.

$$\Delta = \frac{(g*H)^{0.25}*d_{2m}}{\sqrt{Q}}$$

From above equation by putting values of Δ_{low} and Δ_{high} we get d_{2m-low} =109.49mm and $d_{2m-high}$ =92.205mm and d_{2m-avg} =100.85mm.
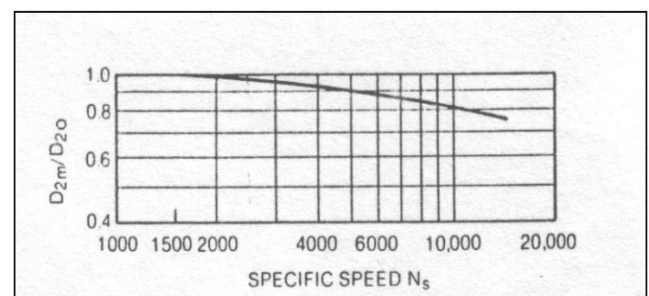Ratio d_{2m} / d_{2o}=0.92 and d_{2o}=109.69mm and d_{2i}= 91.25mm from below figure.

Figure 3.3 d_{2m}/d_{2o} ratio design chart

Then shaft diameter is found by using,

$$d_s = \sqrt[3]{\frac{360000*P_m}{\tau*n}}$$

But power transmitted Pm is given by,

$$P_m = \frac{\rho*g*H*Q*m}{\eta}$$

By taking n=2880 rpm, τ=40 MPa, and Pm=906.13 hp d_s=18mm.

Impeller inlet eye diameter is given by,

$$d_o = K_0 \sqrt[3]{\frac{Q}{n}}$$

K_o value is between 4 to 4.5. for better efficiency take it near to 4. So value of d_o= 55.75mm

The volumetric and mechanical efficiencies are taken as 96% for the pump to be designed. The pump efficiency is taken as 76% we can find hydraulic efficiency from the equation below

$$\eta = \eta_h * \eta_v * \eta_m$$

Using the volumetric efficiency, η_v and the design point flow rate Q, the flow rate Qi can be calculated by using equation below,

$$Q_{i=}\frac{Q}{\eta_v}$$

Q_i=0.0085 m^3/sec

A_0, net area in the impeller inlet eye, is given by,

$$A_o = \frac{\pi(d_o^2 - d_s^2)}{4}$$

A_0 value is 2186.59mm2

Inlet blade angle β_1=25^0 and outlet blade angle β_2=18^0

Geometrical values obtained from analytical calculation are listed below

Table 3.1: Geometrical parameters obtained from analytical design of the impeller

Quantity	Symbol	Value	Unit
Shaft Diameter	d_s	18	Mm
Hub Diameter	d_h	25.2	Mm
Inlet Diameter	D_{2i}	91.25	Mm
Outlet Diameter	D_{2o}	109.6	Mm
Blade Inlet Angle	β_1	25	Degree
Blade Outlet Angle	β_2	18	Degree

3.2. Modeling of Impeller

Solid modelling of the impeller is done using Ansys blade modeller available under Ansys workbench component system Vista CPD. By giving parameters like input and output angle ,tip diameter meridional impeller graphic sketch is available as in below diagram.
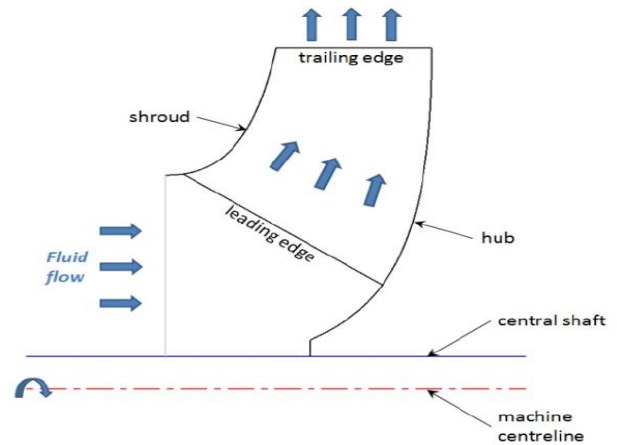

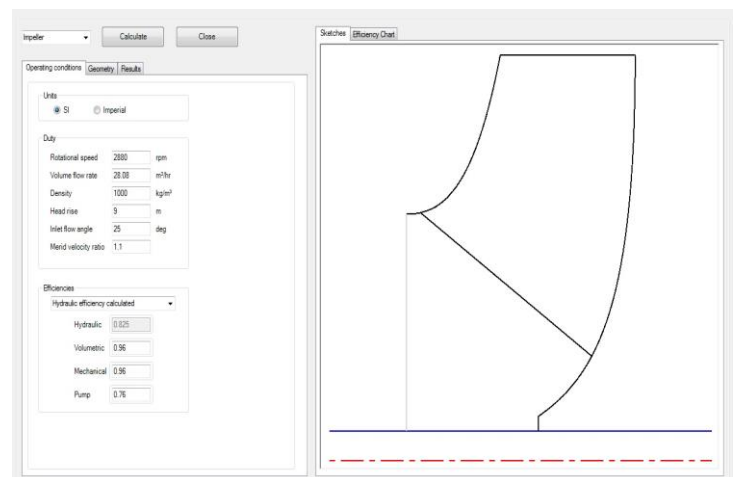Figure 3.4 Meridional Sketch of the Impeller
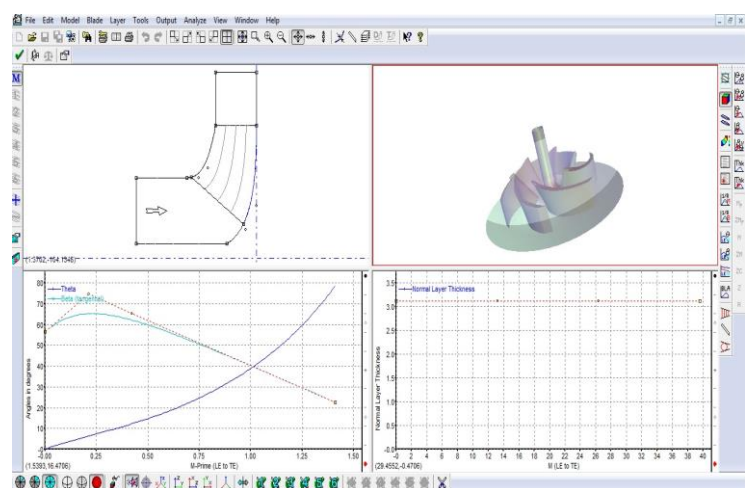

Figure 3.5 Blade modelling in VISTA CPD

Figure 3.6 Impeller model with leading to trailing edge blade profile

4. ANALYSIS OF IMPELLER

4.1. Analysis of Pump Impeller

It follows the following steps.

4.1.1. Solid Modeling

Solid model of the impeller blade is done using ANSYS Bladegen module, and it is shown in Figure 4.1
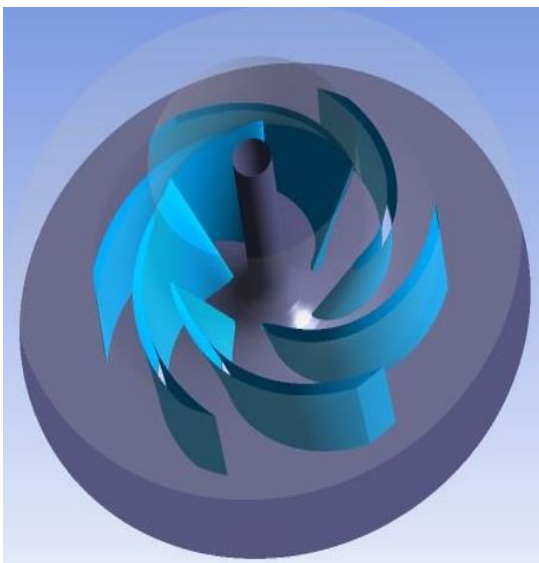

Figure 4.1 Solid Model of Impeller from ANSYS Bladegen

4.1.2. Meshing

Geometry has to be broken up into small, manageable pieces called elements. The corner of each element is called a node, and it is at each node that a calculation is performed. All together these elements and nodes comprise the mesh. The pump impeller meshing is carried out using a ANSYS TurboGrid Meshing tool as shown in the figure 4.2.
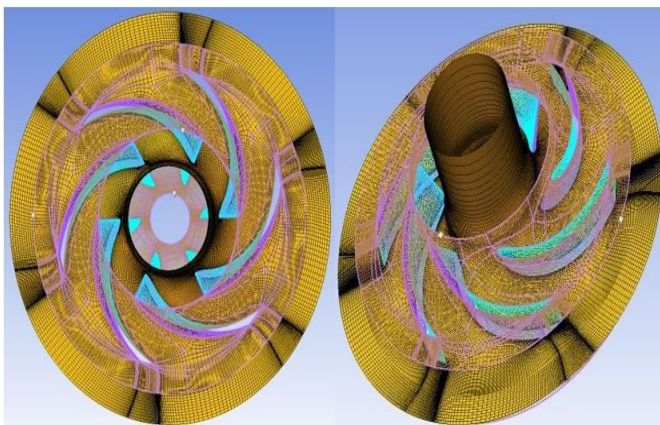

Figure 4.2 Meshing of Impeller using ANSYS TurboGrid.

4.1.3. Material Selection

Volume inside the suction intake, lower and upper extensions are assigned to water and impeller is treated as solid material and torque calculation is done. Impeller surrounding volume is taken as rotating, and speed and axial rotation are mentioned.

4.1.4. Definition of Boundary Condition

At inlet and outlet rise in pressure is determined in analysis. At inlet I atm. Pressure is defined and at outlet volumetric flow rate is defined to determine rise in pressure. Fluid domain is selected in CFX pre basic with material properties of water and shear stress model. After this solver control is defined with 1000 number of iterations.
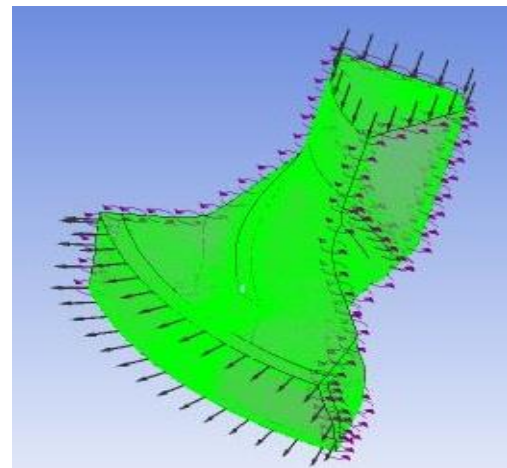

Figure 4.3 Definition of Boundary Conditions using CFX

4.1.5. Analysis Setup

Software automatically selects incompressible when material assigned is water to volume.

4.1.6 Analysis of Pump

CFD analysis results are available at results post. Pressure and velocity are determined across hub and shroud. To found the hydraulic efficiency torque is utilized and head also found as below.
Head is given by below equation,

$$H = \frac{1}{2*g}(V_{f1}^2 + U_2^2 - V_{f2}*cosec^2\beta_2)$$

The values of V_{f1} and V_{f2} are determined by velocity triangles and V_{f1}= 6.416m/sec and V_{f2}= 13.2 and U_2=16.53m/sec and β_2=18 degree and g= 9.81m/sec. then value of head becomes H=8.98m/sec= 9m/sec.

Hydraulic efficiency is given by,

$$\eta_h = \frac{\rho * Q * g * H * m}{\tau * \omega}$$

Where ω=301.59 rad/sec, τ=16.24Nm, m=6, ρ=1000kg/m³ and H=9.

Therefore hydraulic efficiency η_h =87.87%.

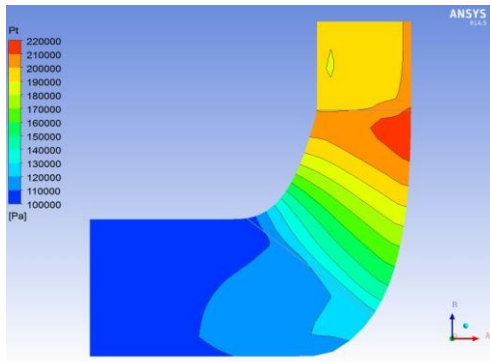The values obtained from CFD analysis are as shown below.

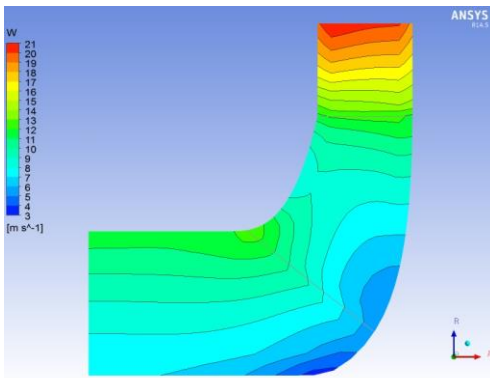Figure 4.4 Contour of Mass Averaged Pt on Meridional Surface

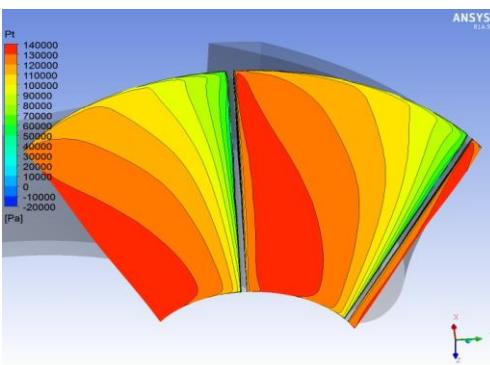Figure 4.5 Contour of Mass Averaged W on Meridional Surface
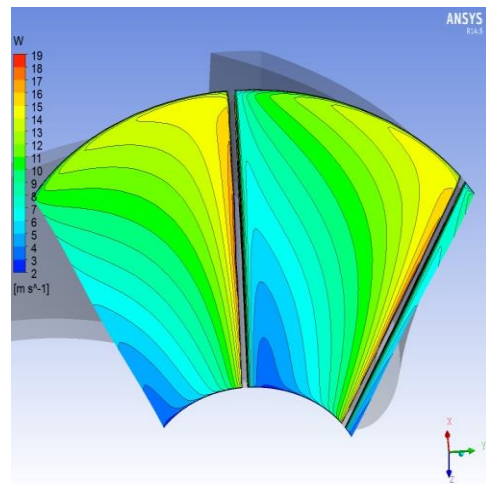

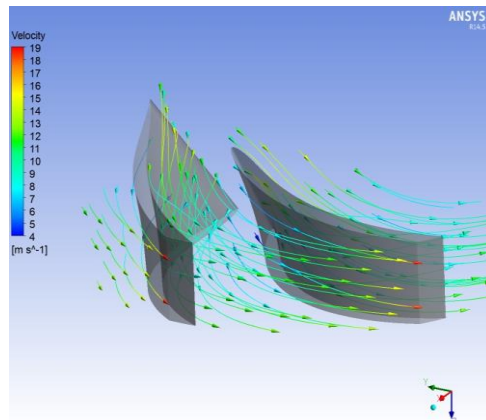Figure 4.6 Contour of Pt at blade LE

Figure 4.7 Contour of W at Blade LE

Figure 4.8 Velocity Streamlines at Blade TE

Table 4.1: CFD Results of pump parameters

Head developed	9.60	M
Discharge	0.008125	m³/s
Hydraulic efficiency	93.68	%
Total efficiency	86.67	%
Shaft power	2127.46	W
Outlet velocity	7.28	m/s

Table 4.2: Comparisons of CFD and analytical results

Parameter	Analytical calculations		CFD results	
	Existing design	**New design**	**Existing design**	**New design**
Head	7.45 m/stage	9 m/stage	8.12 m/stage	9.60 m/stage
Hydraulic efficiency	65%	87.87%	67.1%	93.68%

5. RESULTS AND CONCLUSION

From table 4.2 i.e. the comparison between CFD and analytical results of existing and new designed impeller we can conclude that,

1) Head is increased from 8.12 m/stage to 9.60 m/stage in CFD results, 7.45 m/stage to 9 m/stage in analytical results.

2) Hydraulic efficiency is also increased from 67.1% to 93.68%.in CFD and in analytical results also increased from 65% to 87.87%.

From these points (1) and (2) it can be concluded that, the overall efficiency of the newly designed pump is increased compared to existing pump.

REFERENCES

[1] V.G Chhanya, P.J Thoriya , "*Design And CFD Analysis Of Submersible Pump Impeller Of Mixed Flow Type For Performance Improvement*", International Journal of Advance Engineering and Research Development, Volume 2, Issue 6, June -2015, e-ISSN(O): 2348-4470p-ISSN(P): 2348-6406, pp. 468-475.

[2] Suresh Pittala, Diriba T, "*Computational Fluid Dynamics Analysis of Impeller Design For A Pump*", International Journal of Mechanical Engineering (IJME), Vol. 5, Issue 4, ISSN(P): 2319-2240; ISSN(E): 2319-2259: Jun - Jul 2016; pp. 63-74.

[3] Hao Bing, Shuliang Cao, "*Three-dimensional design method for mixed-flow pump blades with controllable blade wrap angle*", Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 2013 227: 567 originally, DOI: 10.1177/0957650913489296

[4] Jidong Li, Yongzhong Zeng, "*Optimum Design on Impeller Blade of Mixed Flow Pump Based on CFD*", Procedia Engineering 31, 2012, pp. 187-195.

[5] Micha N l Varchola, Peter Hlbocan, "Geometry Design of Mixed Flow Pump Using Experimental Results of on Internal Impeller Flow ", Procedia Engineering 39, 2012, pp. 168-174.