

# To Investigate Effect of Geometrical Parameters on Performance of Low Specific Speed Centrifugal Pump Using CFD

Kunal Gharat<sup>1</sup>, Pawan Gajare<sup>2</sup>, Snehal Murumkar<sup>3</sup>, Parth Gandhi<sup>4</sup>, Amol Mangrulkar<sup>5</sup>

B.E Student (Final year) of Department of Mechanical Engineering, MCTs Rajiv Gandhi Institute of Technology, Mumbai, India<sup>1,2,3,4</sup>

Assistant Professor, Department of Mechanical Engineering, MCTs Rajiv Gandhi Institute of Technology, Mumbai, India<sup>5</sup>

**ABSTRACT:** Effect of blade thickness ratio and geometry of the blade at the outlet of impeller on the performance of low specific speed centrifugal pump was investigated. The type of fluid to be pumped (water) and its properties were determined based on application of pump. Based on this, the type of impeller was decided as closed type impeller. The range of number of vanes was also decided based on fluid properties from literature survey. In CFD, this model was then simulated for operating conditions of low specific speed centrifugal pump. The first parameter blade thickness ratio was varied and its effect on head and efficiency were studied independently. The optimum blade thickness ratio for the given application was determined. The geometry of the blade at the outlet of the blade was varied and change in efficiency was observed. The effect of both these parameters was studied by keeping overall efficiency of the pump as optimization parameter. The results obtained indicated the efficiency increase. The results obtained indicated overall efficiency rise of 4.743% from the standard design of impeller.

**KEYWORDS:** blade thickness ratio, CFD, RANS, vane CPD, ICEM, k- $\epsilon$ , SST

## I. INTRODUCTION

STUDIES by the United States Department of Energy/Lawrence Berkeley National Laboratory and the European Commission show that more than 20% of the global motor electrical energy consumption is consumed by pumps. [1] This illustration shows that pumps consume 22% of the industrial energy consumption, hence the area which has great energy saving potential.

Centrifugal pumps are the rotary machines in which fluid is pressurized or *pumped* majorly due to velocity changes that occur between inlet and outlet. In short, the kinetic energy of the fluid at inlet is converted into pressure energy at outlet. This pressure energy in turn helps in elevating fluid to the desired height.

A centrifugal pump comprises an impeller rotating at a high speed within a stationary casing. The input of centrifugal pump is the mechanical energy imparted by the prime mover through the impeller. In a centrifugal pump the liquid is forced by atmospheric or other pressure into a set of rotating vanes. The parameters affecting the performance of pump are type of impeller, number of vanes, blade exit angle, volute geometry. The flow in a centrifugal pump is very difficult to visualize. Computational fluid dynamics is the analysis of systems involving fluid flow, heat transfer, and associated phenomenon such as chemical reaction by means of computer simulation. Computational Fluid Dynamics is an effective tool that can be used to design machine. The project thus employs CFD as a tool to optimize the design of pump.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 5, May 2017

. The use of CFD-tools to analyze the flow field in turbomachines and to predict performance parameters has gained enormous popularity in recent years. This process has been stimulated by the availability of commercial packages allowing turbomachinery analysis. Hence a CFD model was used for the detailed analysis of the pump. CAD models of a centrifugal pump impeller and volute casing were made on Autodesk Inventor after mean line analysis on Vista CPD. The pump was analyzed on CFX 14.0. based

## II. DESIGN AND ANALYSIS

For the CFD model multiple reference frame and standard k-ε model was used as suggested in literature. [3] The minimum static pressure area appears at the back of blade at impeller inlet region vs. the maximum one at volute tongue and outlet regions. The low-pressure areas are located at the inlet of the impeller and need to be checked so that cavitation does not occur. The Rotor is a working part that adds energy to the fluid; its geometry plays a fundamental role in the centrifugal pump performance. Any change in the rotor geometry would have an effect on the impeller inlet or exit velocity triangles. Hence the blade exit angle has a very important role in the performance of the centrifugal pump. When the outlet angle increases, the centrifugal pump performance handling viscous fluids improves. The efficiency of a twisted-blade impeller will be greater than that of a straight-blade impeller. The pressure increases with increasing radius and the pressure uniformity is affected by the flow rate and the blade exit angle. More pressure uniformity is obtained for less flow rate and less blade exit angle especially in the pump volute region. Similarly, the pump head, the best efficiency and output power are obtained for the highest (open) blade exit angle side. In both cases, low and high flow rates, the pressure at the pump outlet increases with increasing blade exit angle. The blade exit angle has influence on the pump characteristics and performances, especially at high flow rates. Rise in head, hydraulic efficiency and output power have been observed with increasing blade exit angle. Extensive literature survey indicated that a low specific speed centrifugal pump gives maximum efficiency when the exit angle is in the range of 22°-25°. [3] Different cross section shapes for volutes include rectangular, circular, and trapezoidal cross sections. circular and trapezoidal cross sections have higher radial force, at design point and low flow rate. At high flow rate, trapezoidal and circular volutes produce better condition. It is seen that circular cross section volute is the best considering head and efficiency. Volute design with circular cross section and radial diffuser shape provides better hydraulic performance. [5]

Pressure losses in a pump are basically caused by different sources: *Friction* between the fluid passing through different elements of a pump and the walls (friction losses). *Incidence losses* that occur when there is a difference between the flow direction and the blade angles at entrance of an impeller and entrance of diffuser with blade. Other pressure losses caused by separation of boundary layers, back flow, pre-rotation and recirculation losses (hydraulic losses). Theoretical head of centrifugal pump is given by

$$H = [(p_2 - p_1) / \rho g] + [(V_2^2 - V_1^2) / 2g] + [z_2 - z_1] \quad [1]$$

Where p, v, z are pressure, velocity and elevation at the inlet and outlet respectively.

The mechanical efficiency of the pump is given by

$$\eta_m = \frac{gH}{V_{w2}U_2} \quad [1]$$

Where,  $V_{w2}$  is the axial velocity at outlet and  $U_2$  is the blade velocity at outlet

The hydraulic losses and the recirculation losses are quite large and result in reduced overall efficiency of the pump. The hydraulic efficiency is a result of the hydraulic losses in the casing, impellers and diffuser, and reduces the useful head to

$$H = \eta_{th} * H_{th}$$

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 5, May 2017

The hydraulic losses are generated by skin friction and vortex dissipation in all parts of the pump, but especially where the flow is disturbed by leading edges, curves or other irregularities. The losses are very difficult to calculate analytically, and therefore, statistical data have been used to estimate a realistically achievable hydraulic efficiency.<sup>[5]</sup>

With the increase in blade number, the limitation of space between blade and flow stream gets increased. The area of low pressure region at the suction of the blade inlet grows continuously and the static pressure is gradually increasing with the increase in blade number. The uniformity of static pressure distribution at screw section becomes worse and worse, while at diffuser section, it becomes better and better. The impellers with different blade numbers have an obvious low pressure area at the suction side of blade inlet. With the increase of the blade number, total pressure in the region of flow grows continuously. The head of centrifugal pump grows all the time with the increase of blade numbers and total pressure too, but the change in hydraulic efficiency with variation in blade number is little bit complex.<sup>[6]</sup> Considering all the above factors, A low specific speed centrifugal pump was designed for the following specifications

TABLE I  
PUMP SPECIFICATIONS

Rotational Speed	1450 rpm
Volume Flow Rate	280 m <sup>3</sup> /hr
Density	1000 kg/m <sup>3</sup>
Head	20 m
Inlet Flow Angle	90°
Number of Vanes	6
Blade Thickness	0.03 m
Blade Angle	22°

The mean line analysis of the pump was done on Vista CPD. After the analysis impeller and volute were modeled in Autodesk Inventor 2017. It was then imported in Ansys. The Fluid Domain was meshed using Turbo grid and ICEM to get the mesh of sufficiently high resolution. Total number of cells for both the domains was 1476665. Hexahedral elements were used for the impeller mesh in order to get accurate results in a lesser time period. A steady state simulation was done in order to reduce computational time and to obtain approximate results for further analysis. The equations solved are the three-dimensional Reynolds-averaged Navier-Stokes Equations (RANS). The fluid is incompressible. The CFX coupled solver with implicit formulation is used. For the convective terms, high resolution scheme is used. As a turbulence model, the realizable  $k-\epsilon$  model is used in high resolution mode. The shear stress transport (SST) model however produced better results for given conditions. The SST model is a combination of the  $k-\epsilon$  in the free stream. The SST model does not always converge to the solution quickly, so the  $k-\epsilon$  or  $k-\omega$  models are often solved first to give good initial conditions. The inlet pressure is set to be equal to atmospheric pressure and the flow direction normal to the boundary is imposed at the inlet. At the outlet mass flow rate condition is imposed. In order to reduce computational time, the fluid domain in the impeller is made rotationally periodic. Residuals were set to  $1e-5$  for all the mass and momentum and also the turbulence parameters

Pressure distribution in case of impeller is as shown below. The pressure of fluid is almost constant at the suction pressure of  $-4.827e4$

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 5, May 2017

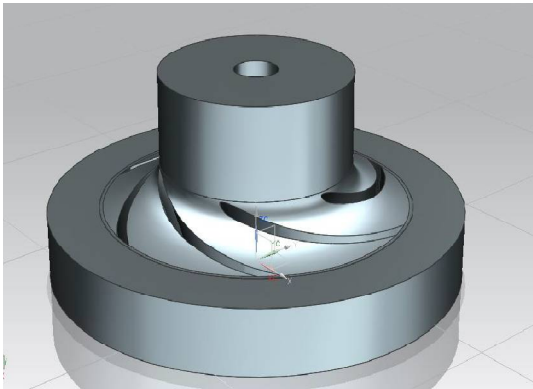


Figure 1 : Model of impeller

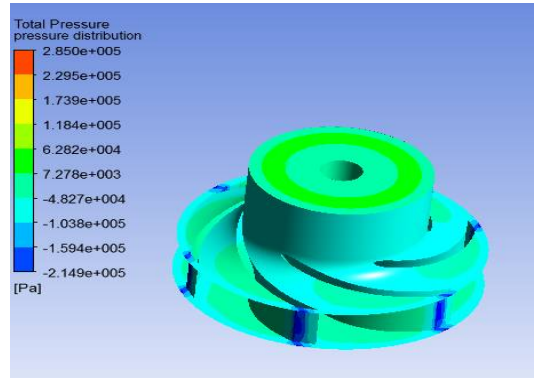


Figure 2: Pressure Distribution in impeller

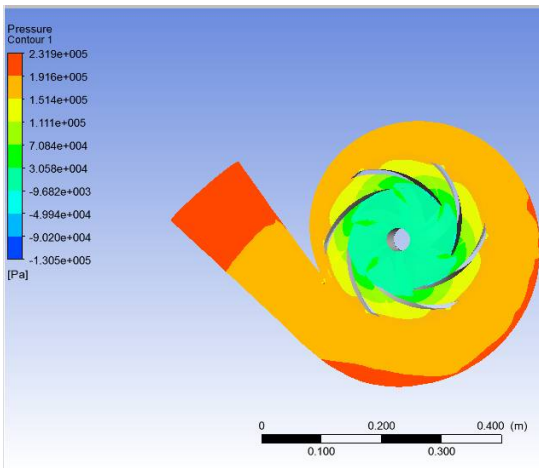


Figure 3: Pressure distribution in involute

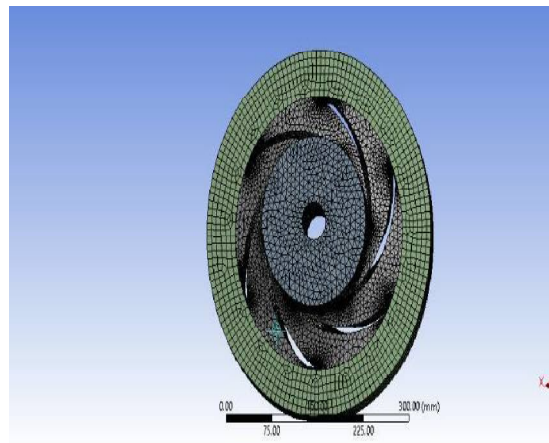


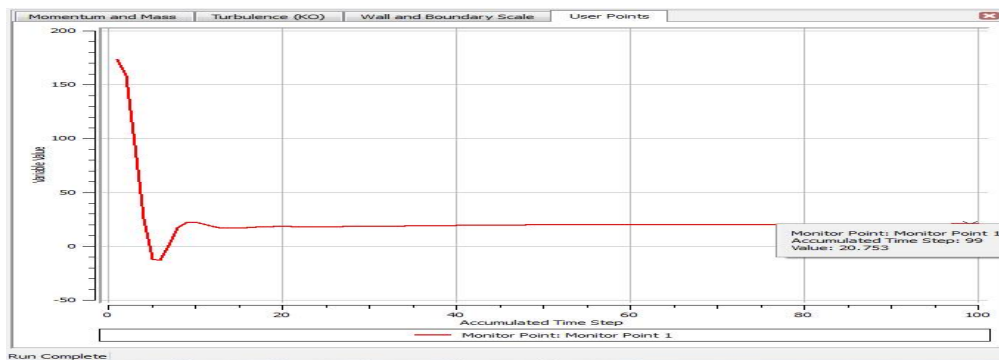
Figure 4: Meshing

## III.RESULTS

### Part 1: Blade thickness ratio

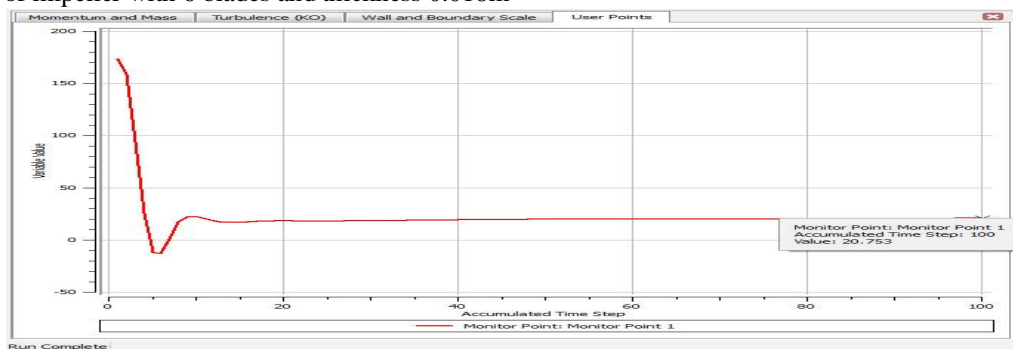
Head curves for the different conditions are as follows

1. Standard design with 6 blades and blade thickness 0.03m gives head as 22m
2. Design of impeller with 6 blades and thickness 0.022m



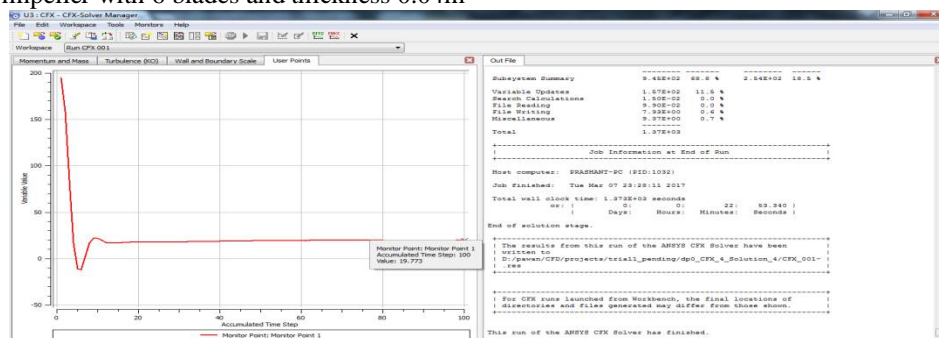
Graph 1

### 3. Design of impeller with 6 blades and thickness 0.018m



Graph 2

### 4. Design of impeller with 6 blades and thickness 0.04m



Graph 3

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 5, May 2017

Thickness	Head(by curve) in meter	Number of iterations	Number of blades	Overall Efficiency (%)
0.04	18.814m	1000	6	73.44
0.03	19.68m	1000	6	76.8
0.022	19.753m	1043	6	77.1
0.018	19.874m	1000	6	77.57

TABLE 2: OBSERVATIONS OF BLADE THICKNESS RATIO VARIATION

## PART 2: FILLET

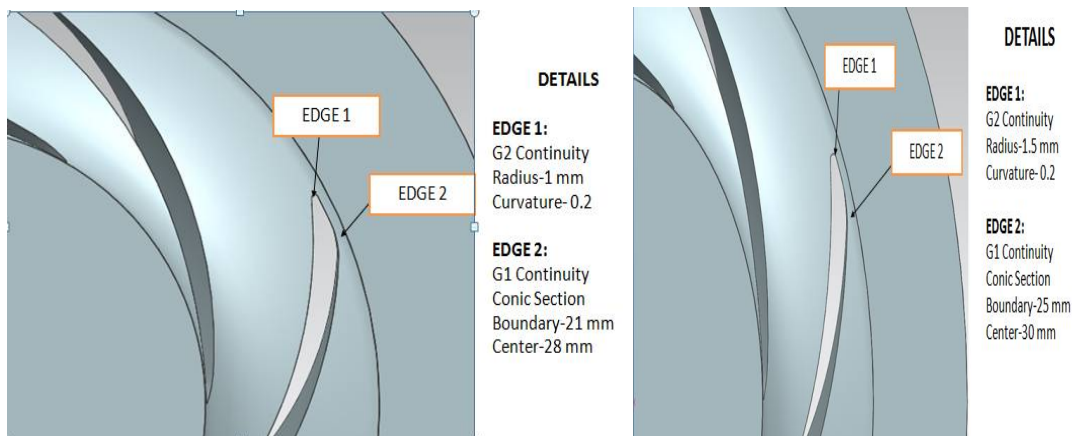


Figure 5: Fillet 1

Figure 6: Fillet 2

Changes in Geometry	Head(m)	Efficiency (%)
Fillet 1	19.983	77.8
Fillet 2	20.898	81.543
Standard design	20	76.8

Table 3: Observations of fillet at outlet

## IV. CONCLUSION

For the selected standard case model, it can be seen that for centrifugal pump, with water as a working fluid the maximum head can be obtained at 0.018m of blade thickness and 6 blades of impeller. But the blade strength reduces therefore we need a much stronger and costlier material. So we selected 0.03 m of blade thickness and 6 blades for performance optimization. Introducing fillet at the trailing edge of blade, head increases significantly to 20.94 m, thereby increasing efficiency. We have used our own cad model and modified the geometry which is quite feasible since it only involves filleting the trailing edge of the impeller and hence we have increased the efficiency of centrifugal pump by 4.743%. Introducing fillet the rise in efficiency can be achieved without any drastic changes in the manufacturing techniques. The changes that are required include change in the mould design



# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 5, May 2017

## REFERENCES

- [1] *A Textbook of Hydraulic Machinery* by R.K.Rajput, S.Chand Publications
- [2] Zhang Shujia, Zhu Baolin, Hu Qingbo, Li Xianhua, "Virtual performance experiment of a centrifugal pump", The MOE Key Laboratory of Mechanical Manufacture and Automation, IEEE (Proceedings of the 16th International Conference on Artificial Reality and Telexistence--Workshops (ICAT'06), 2006.
- [3] Abdelmadjid Chehhat, Mohamed Si-Ameur, "Blade Exit Angle Impact on Turbulent Fluid Flow and Performance of Centrifugal Pump Using CFD", IEEE, 2015.
- [4] Hamed Alemi, Seyyed Ahmad Nourbakhsh, Mehrdad Raisee, Amir Farhad Najafi, "Effects of Volute Curvature on Performance of a Low Specific speed Centrifugal Pump at Design and Off-Design Conditions", ASME Journal of Turbomachinery, APRIL 2015, Vol. 137 / 041009-1
- [5] Nourbakhsh A, Jaumotte A, Hirsch C, Parizi H. "Turbopumps and pumping systems", Springer-Verlag: GmbH; 2007.
- [6] S.Chakraborty and K.M.Pandey, "Numerical Studies on Effects of Blade Number Variations on Performance of centrifugal pump" Journal : IJMET, Volume 6, Issue 8, pp. 105 to 117