Efficiency Improvement of Low Specific Speed Centrifugal Pump by CFD Techniques

Rahul R. Bagde[1], Pramod R. Pachghare[2]

[1]*P.G. Student, Dept. of Mechanical Engineering, Government College of Engineering, Amravati, Maharashtra, India*

[2]*Asst. Professor, Dept. of Mechanical Engineering, Government College of Engineering, Amravati, Maharashtra, India*

Abstract- **In this paper, the main focus is to increase overall efficiency of low specific speed centrifugal pump. This is done by design optimization of volute hydraulic surface. In this paper, use of ICEM CFD 17.0 is shown for Hexa and Tetra meshing of components of low specific speed centrifugal pump. In this paper, CFD analysis of existing centrifugal pump with geometry modifications in the volute to achieve higher efficiency using ANSYS CFX 17.0 is carried out. It is an obvious fact that study on the characteristics of volute is absolutely necessary to improve the performance of low specific speed centrifugal pump. Total 6 iterations by geometry modification are performed for volute. Throat area, Inlet width and Rotation angle volute are taken into consideration. Comparisons of Rectangular volute vs Circular volute and Simulation results vs Experimentation results are shown by Graphical representation. From the paper, it is cleared that there is an improvement in the efficiency, when circular volute is used over rectangular volute.**

Index Terms- **Efficiency, Volute, CFD, ANSYS etc.**

1 INTRODUCTION

A pump, in general may be defined as a machine, when driven from some external source, lifts water or some other liquid from a lower level to a higher level. Or in other words, a pump may also be defined as a machine, which converts mechanical energy into pressure energy. The pump which raises water or a liquid from a lower level to a higher level by the action of a centrifugal force is known as a centrifugal pump.

2. PUMP TYPES

Depending on the direction of the flow at the impeller exit impellers are distinguished as radial, semi-axial and axial. Accordingly the terms radial, semi-axial and axial pumps are used; the latter are also called "propeller pumps". "Closed impellers" are the impellers with front shroud, those without a front shroud are termed "semi-open impellers" and those with large cut-outs in the rear shroud are designated as "open impellers". According to the flow direction at the diffuser inlet there are radial, semi-axial and axial diffusers. Vaneless diffusers are rarely built. The most frequent type of diffusing element for a single-stage pump is a volute. Sometimes there is a concentric annulus or a combination of annulus and volute.

3. LITERATURE REVIEW

E.C.Bacharoudis et al. [1] did the parametric study of a centrifugal pump impeller by varying outlet blade angle. 3 impellers were taken into consideration for the study. All impellers were having same diameter in suction and pressure side, same blade leading edge angle (β1= 14o) and they vary in blade's trailing edge angle (β2= 20o, 30o and 50o resp.). Through this study it has been concluded that when pump operates at design condition or nominal capacity, gain in the head is more than 6% when angle increases from 20o to 50o and ηH decreases by 4.5%. When pump operates at off-design condition, % rise of head curve due to increment of β2 is larger for high flow rates and becomes smaller for Q/Q_N < 0.65 and there is significant improvement in ηH. D. Ekardt [2] performed the detailed accurate measurements of velocities, directions, and fluctuation intensities with a newly developed laser velocity meter in the internal flow field of a radial discharge impeller, running at tip speeds up to 400 m/s. Splitter blade design method was introduced by H. Chen et al [3] for low specific speed centrifugal pump. Splitter blade was set among the long blade to change the velocity and pressure distribution, and the pump performance is improved. It was showed that when the splitter blade is set, the circumferential velocity and pressure becomes even, the vibration and the radiated noise are improved, and the efficiency of the pump can be improved by 1-2% if the blades are set properly. A numerical study had been done by Raul Barrio et al [4] on the pulsating flow at the tongue region for a conventional centrifugal pump with a non-dimensional specific speed of 0.47 and an impeller–tongue gap of 11.4% of the impeller radius. The main focus was the relation between the pressure pulsations and the fluctuating velocity field. The simulations were performed by solving the equations for the modelled pump with the commercial code CFX, after proper analysis of the sensitivity of the results with respect to the mesh size and other numerical parameters. In addition, the predictions of the model were contrasted with experimental data previously collected at laboratory for the real pump. The performance of the impeller had been studied [5] by keeping the outlet diameter same and varying the blade numbers. The main focus of the investigation was the performance characteristics of pump. The methods of numerical simulation and experimental verification were used to investigate the effects of blade number on flow field and performance of a centrifugal pump. 2-D steady numerical analysis was performed for centrifugal pumps with impeller blades 7, 8 and 9. It had been investigated that the efficiency is maximum for 7 bladed impeller centrifugal pumps. In 2015, Hamed Alemi et al. [6] conducted the research on Effects of volute curvature on performance of a low specific speed centrifugal pump at design and off-design condition. In this study, 3 volutes with different geometries and same cross sectional area are taken into consideration. It was seen that circular cross-sectional volute is the best considering head and efficiency. However, circular and trapezoidal cross sectional have higher radial force, at design point and low flow rate. At high flow rate, trapezoidal and circular volutes produce better condition. Hydraulic radii of circular and trapezoidal geometries are higher than rectangular one which causes reduction in pressure loss. In the same research, volute was designed with different diffuser shapes also. There are two alternatives for the selection of diffuser shape in volute design; namely, Radial and Tangential diffuser. Both diffusers provided more or less the same head but radial diffuser gives a slightly higher efficiency at design point and higher flow rate. Also radial diffuser produced lower radial force at design point and low flow rate. It was also concluded in the same research that in low Nq pumps, the generated radial force at high flow rate was significantly more than radial force at low capacity. Mou-Jin Zhang et al. [7] studied the complex 3-D flow field in a centrifugal impeller with low speed. Coupled with high Reynolds number k-ἐ turbulence model, the fully 3-D Reynolds averaged Navier-Stokes equations were solved.

4. CALCULATION OF MAIN DIMENSIONS

IMPELLER:

Inlet Triangle:

The meridional velocity immediately in front of the impeller blade leading edges is $C_{1m} = Q_{La}/A1$, where A1 is calculated in accordance with the position of the leading edge. Immediately after the leading edges the meridional velocity is increased to $C_{1m}' = \tau \times C_{1m}$ due to the blade blockage. The circumferential components of the absolute or relative velocity are not affected by the blockage, as follows from the conservation of angular momentum. The fluid flow to the impeller is mostly axial ($\alpha 1$ = 90°). The circumferential component of the absolute inflow velocity is therefore $c1u = 0$. It can be seen that the approach flow angle $\beta 1$ of the blades is increased by pre-swirl and reduced by counter-swirl. As a result of the blade blockage c1m grows to c1m′ so that the approach flow angle increases from $\beta 1$ to $\beta 1'$. The difference between blade angle $\beta 1B$ and flow angle $\beta 1'$ is known as incidence: $i1' = \beta 1B - \beta 1'$. If the incidence is zero, the blade has only a displacement effect on the flow; local excess velocities are correspondingly low. They grow with increasing incidence until the flow separates, since incident flow generates a circulation around the leading edge at $i1 \neq 0$. With a certain flow rate (i.e. a specific c1m') blade and flow angles are identical ($\beta 1B = \beta 1'$) and the incidence becomes zero. This flow situation, called "Shockless Entry", is calculated from the condition $\tan \beta 1' = \tan \beta 1B$, where $\phi 1$, $SF = c1m'/u1$ is the flow coefficient. If the approach flow angle drops below the blade angle (i1 > 0), the stagnation point is situated on the pressure surface of the blade. If the pump flow rate exceeds the value of the shockless entry, the incidence is negative and the stagnation point is located on the blade suction surface.

Outlet Triangle:
Blade blockage is still present immediately upstream of the impeller outlet and the velocity is correspondingly greater than downstream of the trailing edge: c2m' = c2m × τ2. Again, the blockage does not affect the circumferential component. The absolute velocity c2 and outflow angle α2 are relevant for the design of the diffusing elements.

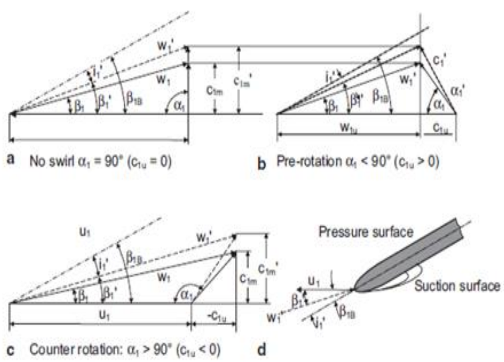

Fig.2.1 Inlet Velocity Triangles

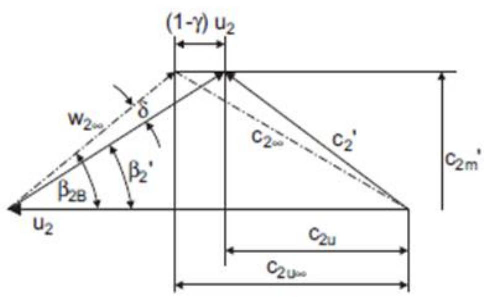

Fig.2.2 Outlet Velocity Triangle

VOLUTE:
In the volute the kinetic energy present at the impeller outlet is converted into static pressure with as few losses as possible. The volute then directs the fluid into the discharge nozzle or, in multistage pumps, into the following stage. Prior to calculating a volute, the boundary conditions for the casing design must be established. In particular it must be decided whether a single or double volute is required. Volute can be categorized as Single volute, Double volute and Twin volute.

Wrap Angle of Partial Volute (ε_{sp}):
In the case of double or twin volutes the wrap angle of the partial volutes generally amounts to $\varepsilon_{sp} = 180°$. In this case an even impeller blade number must be avoided in order to reduce pressure pulsations. If nevertheless a 6-bladed impeller (for example) is to be employed it is advisable to reduce the wrap angle of the inner volute ε_{sp} to 165° or 170° so that two blades never pass the cutwaters simultaneously. A similar procedure should be employed in the case of multiple volutes. The selection of a wrap angle of 165° may reduce the risk of flow instabilities and alternating stall. This design option could be recommended for general use since the increase in static radial thrust as compared to 180° is moderate and a defined radial force might be advantageous with respect to bearing design and vibrations. In the case of axial-split pumps wrap angles of less than 180° are sometimes selected to avoid that the centre rib passes through the split flange of the casing (matching difficult because of casting inaccuracies). Double volutes with wrap angles under 90° must not be used since they are unable to reduce the radial forces. Wrap angles over 180° should be avoided as a matter of principle since the long outer channel causes additional losses, and high radial thrusts must be expected at q* > 1.

Casing Design Flow Rate (Q_{Le}):
To ensure that the actual best efficiency flow coincides with the design flow rate, the volute must be designed for Q_{opt}. The design flow rate must be augmented by possible leakages which may flow through the volute. Note that the leakage losses at the impeller inlet do not flow through the volute.

Inlet Velocity (c_{3u}):
Downstream of the impeller, circumferential component of the absolute velocity develops in accordance with the conservation of angular momentum as per $c_{3u} = c_{2u} \times r_2/r_3$. With some types of pumps a diffuser or stay vanes are arranged between the impeller and the volute. In this case the circumferential velocity c_{4u} at the outlet of these components must be used as inlet velocity to the volute.

Cutwater Diameter (d_z^*):

Between impeller and cutwater a minimum clearance must be maintained to limit pressure pulsations and hydraulic excitation forces to allowable levels. The ratio of the cutwater diameter $d_z^* = d_z/d_2$ is calculated from the formula as a function of n_q and H_{st}.

Shape of the Volute Cross Sections:

The shapes must be selected so as to suit the pump type, taking into account casing stresses and deformations as applicable. When designing the casing, also the requirements of economic manufacture of patterns and castings must be taken into account. With the rectangular and trapezoidal basic shapes, for instance, all corners must be well rounded for casting reasons and flat surfaces designed with a slope ("draft angle") in accordance with the pattern split plane. Concrete volutes have specific shapes to facilitate shuttering. Rectangular and trapezoidal shapes offer the advantage to be developed on surfaces of revolution, something that facilitates design and manufacture. Volutes with circular cross sections that cannot be arranged on surfaces of revolution are employed, for instance, when the volute is welded together from segments. Semi-axial impellers may have asymmetric cross sections. In double volutes the cross sections of the inner volute and the outer channel must be matched so that the casing outer wall is given an easily producible shape, and a simple transition to the circular discharge nozzle is achieved. Basically, the designer has considerable freedom to configure the cross section without risking major losses in efficiency. It is expected that flat cross sections (similar to flat elbows) result in less intense secondary flow than circular cross sections, thereby generating fewer losses. For this reason, flat cross sections with a ratio of width B to height H in the range of B/H = 2 to 3 may be considered the optimum.

Inlet Width (b_3):

The inlet width is obtained from the impeller outlet width b_2, the necessary width of the impeller sidewall gaps and requirements of the casing design. In particular the transition from the volute to the discharge nozzle should be smooth. This requirement means a relatively large b_3 in single volutes so that the ratio h/b at the volute throat area is close to 1.0. In a double volute the ratio h/b is rather near 0.5. The ratio b_3/b_2 depends on the specific speed; it can be selected in wide limits without major effect on the efficiency. Open impeller sidewall gaps are favourable with respect to efficiency and radial thrust. At small specific speeds they result in values of b_3/b_2 = 2.0 to 4.0. In some applications vibration problems were reported on pumps with impeller sidewall gaps wide open to the volute. However, large values of b_3/b_2 are prohibited for design reasons at high specific speeds and relatively wide impellers; b_3/b_2 = 1.05 to 1.2 may then be selected. At high nq large ratios of b_3/b_2 are unfavourable because strong secondary flows and turbulent dissipation losses would result.

Volute Throat Area:

Volutes are mainly designed in accordance with the theorem of angular momentum conservation. Different approaches to the volute design did not produce any measurable improvements in efficiency. A partial volute with the wrap angle ε_{sp} must be designed for the flow $Q_{Le} \times \varepsilon_{sp}/2\pi$ where $\varepsilon_{sp} = 2\pi$ applies to single volutes ($z_{Le} = 1$). $\varepsilon_{sp} = \pi$ is used for twin volutes or double volutes ($z_{Le} = 2$) with 2 × 180° and $\varepsilon_{sp} = 2/3\pi$ for a triple volute ($z_{Le} = 3$). If all partial volutes have the same wrap angle, $\varepsilon_{sp}/2\pi = 1/z_{Le}$ applies. The throat area of each partial volute must satisfy the equation which is obtained from:

$$\int_{r_z'}^{r_A} \frac{b}{r}\,dr = \frac{Q_{Le}\,\varepsilon_{sp}}{2\pi\, c_{2u}\, r_2}$$

Although the thickness of the volute cutwater causes local accelerations and excess velocities, it has only a minor impact on the best efficiency point flow rate. Integration therefore proceeds from the stagnation point radius $r_z' \approx r_z + e_3/2$ to the outer limitation of the volute cross section at the radius r_a, without taking into account the cutwater thickness.

An analytical solution can also be given for volutes with circular cross sections. The diameter of a circular throat area d_{3q} can serve as an approximation for the rapid estimation and assessment of cross sections of any shape if considered as an equivalent cross section giving the actual throat area of the casing. Large hydraulic losses which occur downstream of the volute in the diffuser shift the best efficiency point to smaller flow rates than would be obtained according to the conservation of angular momentum. In the case of double and twin volutes with low specific speeds the volute must therefore be designed for a slightly higher flow rate by substituting, for instance, $Q_{Le} = (1.05 \text{ to } 1.25) \times Q_{opt}$.

5.MAJOR DIMENSIONS FOR THE PUMP RELATED TO THE PROJECT WORK

Title	Value
Specific Speed $n_{q\,(BEP)}$	10
Q_{BEP}	12 (m^3/hr)
H_{BEP}	17.0 (m of H_2O)
η_{BEP}	1450 rpm

6. RESULT AND DISCUSSION

ANSYS CFX 17.0 SILULATIONS:

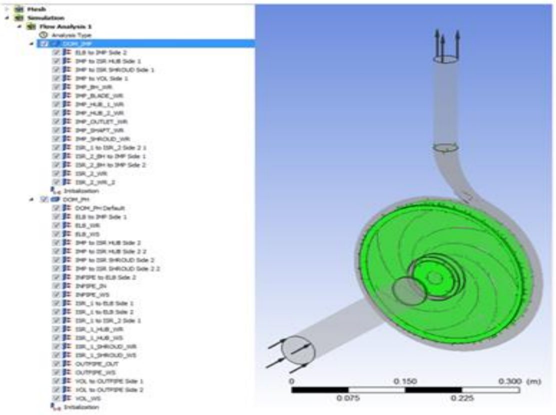

Fig. 4.1 Rotating Parts

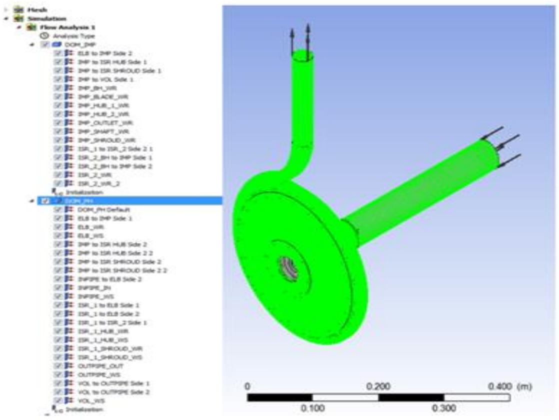

Fig. 4.2 Stationary Parts

BOUNDARY CONDITIONS:

A. Boundary: INPIPE_IN

Basic Settings: Boundary Type- Inlet

Location- INPIPE_IN

Boundary Details: Flow Regime: Option- Subsonic

Mass and Momentum: Option- Mass flow rate

Mass flow rate- Q*dens

Mass flow rate area- As specified

Flow Direction: Option- Normal to boundary condition

Turbulence: Option- Medium (Intensity= 5%)

B. Boundary: OUTPIPE_OUT

Basic Settings: Boundary Type- Outlet

Location- OUTPIPE_OUT

Boundary Details: Flow Regime: Option- Subsonic

Mass and Momentum: Option- Average static pressure

Relative pressure- Discharge pressure

Pressure profile blend- 0.05

Pressure Averaging: Option- Average over whole outlet

CONTOURS:

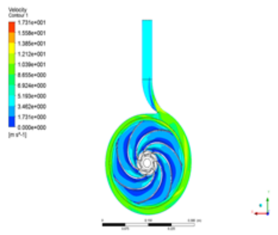

Fig. 4.3 Velocity Contour Global

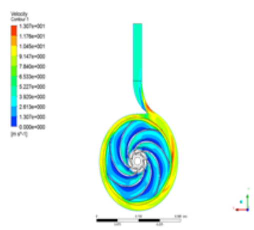

Fig. 4.4 Velocity Contour Local

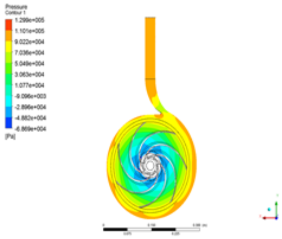

Fig. 4.5 Pressure Contour Global

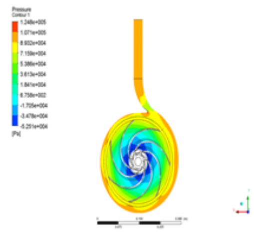

Fig. 4.6 Pressure Contour Local

ITERATION V_01

Iteration Name: V_01

Throat area (A3q) = 303.08 mm2

Inlet width (b3) = 20.8 mm

Rotation angle volute = 0o

Sr. No.	% of Total Discharge	Total Discharge (m^3/hr)	Head (m)	Efficiency (%)	P_2 (W)
1	60	7.2	18.53	38.96	929.72
2	80	9.6	18.00	44.62	1051.11
3	90	10.8	17.53	46.25	1111.08
4	100	12	16.96	47.18	1175.4
5	110	13.2	16.29	47.00	1237.27
6	125	15	15.02	46.21	1322.88

Table 4.1 Simulation Readings for Iteration V_01

Serial No.	Discharge (m^3/hr)	Head (m)	Efficiency (%)	Power (W)
1	1.23	18.99	8.88	716.94
2	1.74	19.09	12.27	738.00
3	2.28	19.16	15.54	764.91
4	3.03	19.27	19.83	800.86
5	5.73	19.24	32.42	925.82
6	6.78	19.00	36.05	973.05
7	7.66	18.71	38.49	1014.17
8	8.97	18.17	41.46	1070.22
9	9.86	17.73	42.90	1109.18
10	10.56	17.35	43.82	1138.16
11	10.97	17.13	44.26	1155.34
12	11.66	16.70	44.76	1184.34
13	12.00	16.55	45.04	1195.07
14	12.35	16.25	45.25	1206.88
15	12.61	16.06	45.28	1217.19
16	13.07	15.72	45.25	1235.97
17	13.58	15.29	45.11	1253.17
18	14.43	14.52	44.47	1282.48
19	15.31	13.67	43.36	1313.80
20	16.26	12.86	42.37	1343.85
21	17.68	11.33	39.29	1387.84
22	18.89	9.14	32.97	1424.42
23	19.75	7.48	27.73	1449.43
24	20.56	5.73	21.78	1471.15

Table 4.2 Experimentation Readings

7. CONCLUSION

The objective set for the project is successfully completed. From the literature survey, it is clear that study on the characteristics of the volute is absolutely necessary to improve the performance of centrifugal pumps. In this project, CFD analysis of existing centrifugal pump with geometry modifications in the volute to achieve higher efficiency using ANSYS CFX 17.0 is done. From the simulation results, following points are concluded.

- Overall Efficiency increases by 1.98% when circular volute is used instead of rectangular volute.
- Required Power decreases by 2.46% when circular volute is used instead of rectangular volute.

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, 2, 75-83, 2008.

[2] D. Ekardt, "Detailed Flow Investigations within a High Speed Centrifugal Compressor Impeller", ASME Journal of Fluids Engineering, 98, 390-402, 2014.

[3] H. Chen, W. Liu, W. Jian and P. Wei, "Impellers of Low Specific Speed Centrifugal Pump based on the Draughting Technology", IOP Conference Series: Earth and Environmental Science, 12, 1-7, 2010.

[4] Raul Barrio, Jorge Parrondo, Eduardo Blanco, "Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points", ELSEVIER Computers and Fluids, 39, 859-870, 2010.

[5] Vijaypratap R. Singh, M. J. Zinzuvadia and Saurin M. Sheth, "Parametric Study and Design Optimization of Centrifugal Pump Impeller- A Review", Journal of Engineering Research and Applications, 4(1), 216-220, January 2014.

[6] Hamed Alemi, Sayyed Ahmad Nourbakhsh, Mehrdad Raisee and Amir Farhad Najafi, "Effects of Volute Curvature on Performance of a Low Specific Speed Centrifugal Pump at Design and Off-Design Conditions", Journal of Turbomachinery, 137, April 2015.

[7] Mou-Jin Zhang, Chuan-Gang Gu And Yong-Miao Miao, "Numerical Study of the Internal Flow Field of a Centrifugal Impeller", ASME (Presented at the International Gas Turbine and Aeroengine Congress and Exposition), 94-GT-357, June 13-16, 1994.