Design, Development and Analysis of Impeller for Sand Slurry Pumping

S. A. Patil
PG Student
DKTES Textile and Engineering Institute, Ichalkaranji, India

V. D. Shinde
Professor
DKTES Textile and Engineering Institute, Ichalkaranji, India

Abstract

The centrifugal pump is one of the most used system in the different types industries and various locations as like municipal water circulation, municipal sewage system, pumping river sand etc. This paper deals with the design of impeller of centrifugal pump, where the flowing medium is the mixture of water 60% by volume and sand is 40% by volume. The design calculations of enclosed impeller are done based on volume flow rate, density of sand water mixture, total head required and pump speed. Based on these parameters the impeller is designed. The 3-D model is generated through the BLADEGEN module of ANSYS 16.0 workbench and the analysis is conducted with the water as flowing medium in fluid flow CFX module. The performance for the solid-liquid mixture by head ratio is predicted accordingly. After predictions results are analysed for total head produced and efficiency of the pump.

Keywords: Impeller, centrifugal pump, sand slurry, head ratio

I. Introduction

In centrifugal pump the flowing fluid is entering through the inlet of the pump, pump receives the kinetic energy from the rotating impeller; this energy accelerates the fluid to a high velocity and then fluid goes through casing to discharge at high pressure. The centrifugal pump design is complex when there is design for the slurries or mixture applications, the realistic situations are not to be expressed well in the design; hence the design or modification is also done by experience in different fields applications. Solid-liquid mixtures types of centrifugal pumps are lower the cost of the system than transportation by manual or conveyor belt.

Design of the centrifugal pump for transporting of slurry; the physical characteristics are dependent on many factors such as size of the particles, concentration of solids in the liquid etc. After design performance test is done on flowing fluid as water and then it predictions done on the different factors for the different applications. Because of the analysis software's there is time and cost saving in the design, modification, testing and manufacturing of the pump for the different application.

II. Design of Enclosed Impeller [1],[2],[6]

For the design, properties of the solid-liquid mixture are taken as fallow:

- Volume flow rate. Q = 0.06 m^3/sec
- Total head. H = 20 M
- Density of the mixture. ρ = 1287 kg/m^3
- Gravitational acceleration. g = 9.81m/s^2
- Pump speed. n = 2000 rpm

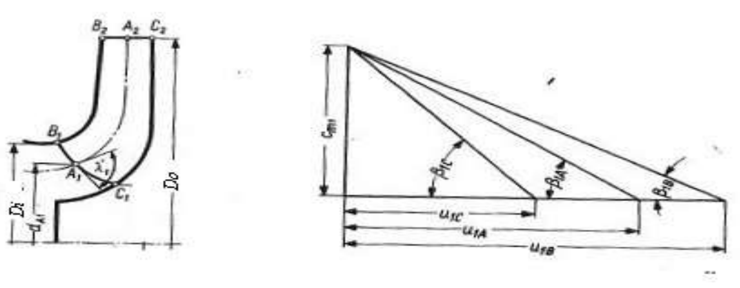

Fig. 1: Constructional diagram of the enclosed impeller [2]

Design parameters are as follow:

- The kinematic specific speed. $n_{sQ} = \frac{n*\sqrt{Q}}{H^{\frac{3}{4}}}$ =51.8

- To determine the hub diameter Calculating shaft input power P_{sh} with assuming the pump efficiency η_p =80%. $P_{sh} = \frac{\rho * H * Q}{75 * \eta} =$ 24.94 metric H.P
- Allowing a reserve 5 % ,the power supplied by drive shaft .P= 26.1 metric H.P
- Calculate the shaft diameter .$d_{sh} = \sqrt[3]{\frac{3600000*P}{\tau * \eta_p}}$ =2.18 cm ≈ 25mm
- The hub diameter d_h = 1.4 * d_{sh} =35mm
- Volumetric efficiency to be considered for design, Assume that volumetric efficiency η_V =93%.Hence, $Q' = \frac{Q}{\eta_V}$ = 0.065 m^3/sec
- impeller inlet diameter(d_o)

The velocity Cm1 = kcm1*$\sqrt{2 * g * H}$ = 4.36 m/sec
From below graph Kcm1 = 0.21 for n_{sQ} =51.8

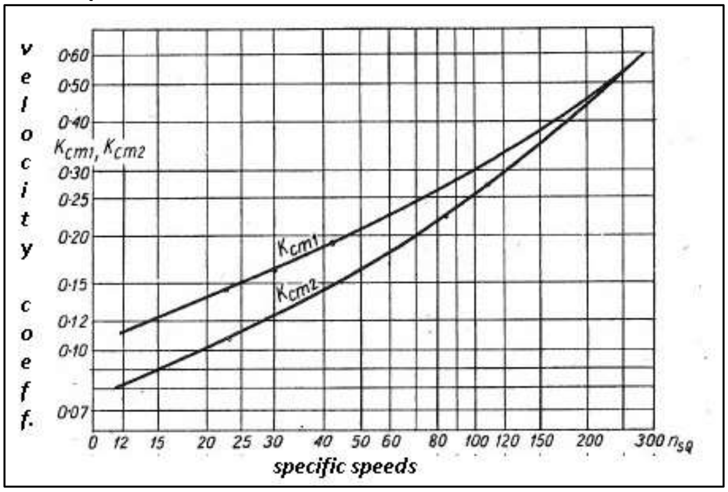

Fig. 2: correlation of velocity coefficient Kcm1 and Kcm2 in relation to the specific speed n_{sQ}[2]

In view of the stable pressure characteristics, we make the blades of double curvature. For this type of blade $C_o = \frac{Cm1}{\varphi 1}$, assume an inlet constriction coefficient $\varphi 1$=1.25, and after calculation check the assumed value is correct or not and if value is not correct change the value and all parameters are calculated.

C_o = 3.45 m/sec

A_o (free inlet area) = $\frac{Q'}{CO}$ = 0.0188 m^2

The diameter of hub d_h =35mm; the cross sectional area of the hub A_h= 0.00096 m^2

A' (total inlet cross sectional area) = A_o+ A_h = 0.0197 m^2

Hence $Di = \sqrt{\frac{4*A'}{\pi}}$ = 0.158 m ≈158mm

- The diameter of the central streamline at inlet $d_{A1} = \sqrt{\frac{do2+dh2}{2}}$ = 0.114m ≈ 114 mm
- Assume the number of blades z =3 and blade thickness s =15mm
- Calculation of blade inlet angle($\beta 1$)

The peripheral velocity at the point A_1 is $u_{1A} = \frac{\pi * Da * n}{60}$ =11.93 m/sec

The inclination angle β_1 of streamlines A_1A_2 , β_1 = $\tan^{-1}\frac{cm1}{u1}$ = 20°

- Checking the assumed inlet constriction coefficient is correct, using following eq^n. $\frac{1}{\varphi 1} = 1 - \frac{s}{t_1} * \sqrt{1 + \frac{cot^2\beta_1'}{sin^2\lambda_1'}}$ =0.67

Where s=15mm, $t_1 = \frac{\pi * d_A}{z}$=119.3, λ_1' =78°, $sin^2\lambda_1' = 0.95$, $cot^2\beta_1$=5.55,$\varphi 1$=1.49

The value of $\varphi 1$=1.49 i.e. it is greater than the assumed value $\varphi 1$=1.25, hence it is necessary to make an adjustment, Now, assume $\varphi 1$=1.47

After making the correction, find that C_o=2.96m/sec, A_o=0.022m^2, A'=0.023m^2, Di=170 mm, d_{A1}=122mm, u_{1A}=12.79 β_1=19°

- calculation the β_1 for streamlines B_1 B_2 and C_1 C_2

$u_{1B} = \frac{3.14*.170*2000}{60}$=17.8 m/sec, $\tan\beta_{1B} = \frac{cm1}{u1b}$, β_{1B}=13°

$u_{1C} = \frac{3.14*.065*2000}{60}$=6.84 m/sec, $\tan\beta_{1C} = \frac{cm1}{u1c}$, β_{1C}= 32°

- Calculation of outlet diameter of the impeller (D_o)

Assume η_h (Hydraulic efficiency) $= \frac{\eta}{\eta_v * \eta_m}$ =93%, H_{th} (theoretical head) = $\frac{H}{\eta_h}$ = 22.2 M
For calculation of u2, value of $\beta 2$ and $1 + Cp$ to be assumed and after calculation assumed value checked and adjustment in to the value must be done.
Assume β_2=20^o, $1 + Cp$ =1.7

The velocity $cm_2 = Kcm_2 * \sqrt{2gh}$ =3.17, from fig.2 Kcm_2 = 0.16 for n_{sQ} =51.8

$$u_2 = \frac{cm2}{2tan\beta 2} + \sqrt{\left(\frac{cm2}{2tan\beta 2}\right)^2 + g * \mathrm{Hth} * (1 + Cp)} = 23.8$$

$$Do = \frac{60 * u2}{\pi n} = 227 \text{ mm}$$

The assumed value of 1+ Cp now be checked by computing the static moment for the central streamline, $Cp = \frac{r_2^2 * \psi}{z * \mathrm{Mst}}$ = 0.66 , This value of Cp is closed to assumed value of 0.7 and so it is not necessary to make adjustment to the $\beta 2$and d_2

– Outlet width of the impeller (b_2)

There is practical or on field limitations hence b_2=60mm

III. Development of enclosed impeller

By using the above design calculations, 3-D model is constructed using the BladeGen component system which present in ANSYS 16.0(workbench).

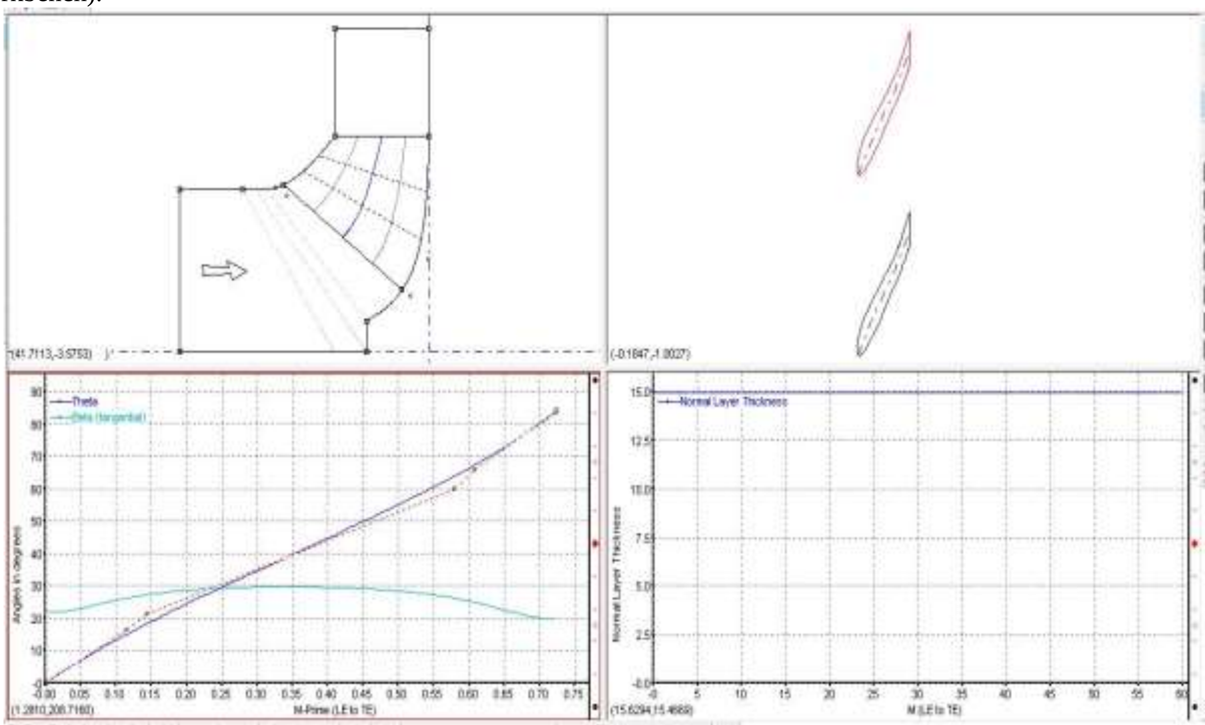

Fig. 3: BladeGen component system

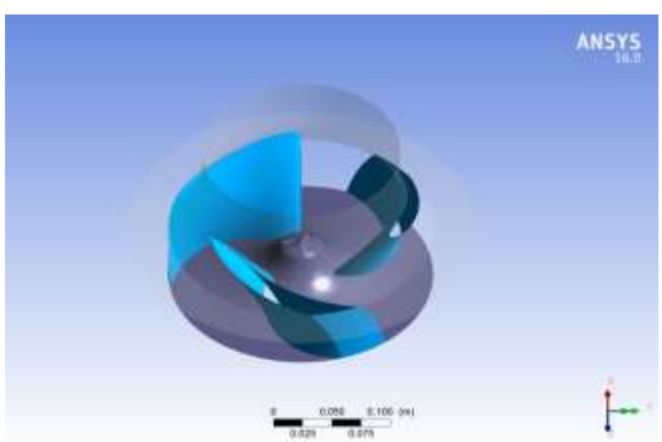

Fig. 4: 3-D Model of the enclosed impeller for CFX

IV. CFD ANALYSIS OF ENCLOSED IMPELLER[4],[7],[8],[9]

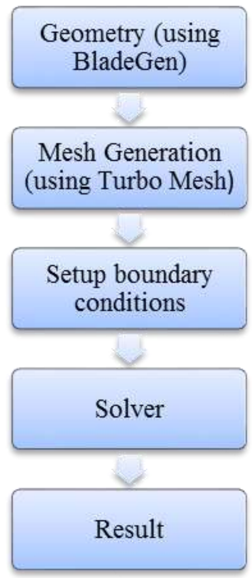

Fig. 5: CFD Methodology

Generated model of impeller is transferred to the Turomachinery fluid flow for analysis

A. Mesh generation

In Turbo Mesh edit mesh data. In the details view, set size factor to 1.0. Hence Total Nodes=424160, Total Elements=394010

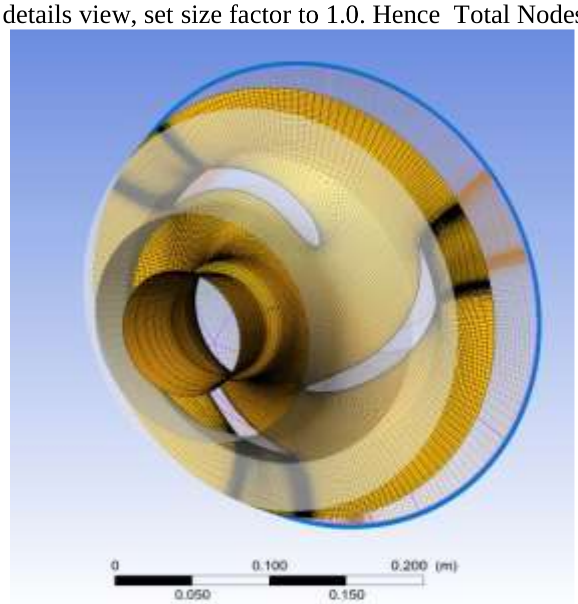

Fig. 6: Mesh generated in Turbo Mesh

B. Setup Conditions

In Turbo Mode setup the boundary conditions. By accepting the default settings, next and give the pump speed 2000rpm. Next step > in the physics definition panel. Select fluid is water; turbulence model is shear stress transport. Give outflow mass flow rate per machine is 60 kg/s.

In next step checking the interface has been setup. Next step by checking boundaries created in the boundary definition panel. Proper location of the inlet, outlet, blade, hub, hub outlet and shroud are to be given. In the location of the Blade, Hub, Hub outlet and shroud; Boundary type is 'wall' and wall influence is 'no sleep wall'. In location of inlet Boundary type is 'inlet' and in flow specification option 'stationary frame total pressure' is selected and relative pressure set to the '0 atm'.after next finish the operation and going to the general mode. In the outline tree view edit the 'Outlet' and give the mass flow rate 60kg/sec.

1) Finish the setup

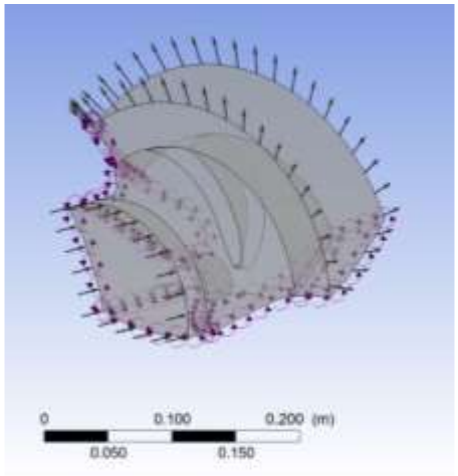

Fig. 8: Representation of impeller part after boundary conditions are to applied

In solver control criteria defining the following details
No. of outer loop iteration = 100, Convergence criteria-residual target is= $1e^{-4}$

C. Solver

The CFX-SOLVER is solving problem and finish after some time.

D. Result

In result cell, the following result is generating 'PUMP IMPELLER REPORT'.

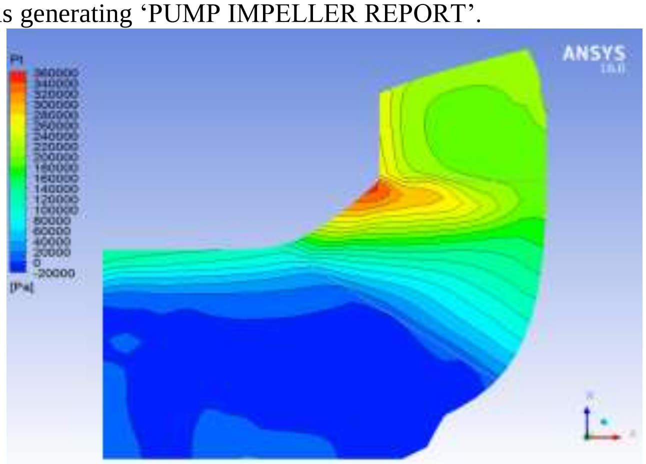

Fig. 9: Pressure distribution

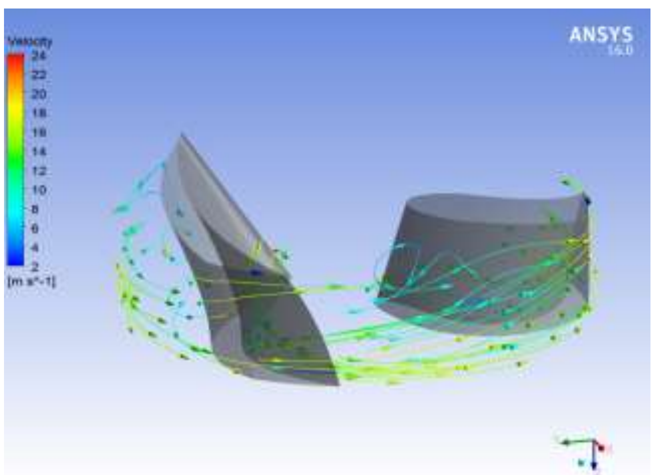

Fig. 10: velocity Streamlines at Blade trailing edge

Table – 1
Performance results

Rotation Speed - rpm	*2000*
Reference Diameter - m	*0.227*
Volume Flow Rate – m^3/s	*0.0605*
Head (LE-TE) - m	*27.4940*
Head (IN-OUT)- m	*25.5942*
Shaft Power W	*18555.8000*
Total Efficiency (IN-OUT) %	*81.6140*

Result are to be consider for the sand and water mixture and correction in head and efficiency
Sand % in mixture 40% (by volume), Sand % in mixture 52% (by weight) = C_w [5]
Specific gravity of sand = 1.6 = S [10], Average diameter of the sand = 1 mm = d_{50}
HR (Head Ratio) [3] = $1-0.0385*(S-1)*(1+\frac{4}{S})*C_{w*} \ln(\frac{d_{50}}{0.0277})$ = .85
HR = ER (Efficiency Ratio) Hence
Total Head for the solid-liquid mixture = .85 * 25.5942= 21.75 m
Efficiency for the solid-liquid mixture = .85 * .816140= 0.693= 69.3 %

V. Conclusion

The following conclusion can be drawn based on the above calculations and analysis of the enclosed impeller

1) The CFD results show satisfactory with the design calculations.
2) The impeller design is created for solid- liquid mixture, and further analysed by ANSYS CFX with the flowing fluid medium is water. The total head observed 25.5942 M for water and 21.75M for the sand and water mixture.
3) The total efficiency of pump was analysed by ANSYS CFX was found to be 81.61% and predicted 69.3% total efficiency for the solid-liquid mixture.

References

[1] K. C. Thin., M. M. Khaing., K. M. Aye., (2008), "Design and Performance Analysis of Centrifugal Pump" World Academy of Science, Engineering and Technology, Vol. 46, 422-429.
[2] S. Lazarkiewicz., (1965) "Impeller Pumps" Wydawnictwa Naukowo-Techniczne Warszawa, 154-180.
[3] H. P. Cheng., S. P. Yun., (2013), "A Simple Correlation Equation to Predict Pump Performance for Slurry" Applied Mechanics and Materials, Vol. 365-366, 365-366.
[4] L. Wai, W. Shi, X. Jiang, B. Chen, Y. Wu., (2012), "Analysis on Internal Solid-Liquid Two-phase Flow in the Impellers of Sewage Pump" International Conference on Advances in Computational Modeling and Simulation, Vol.31, 170-175.
[5] G. Wilson., "The Effects of Slurries on Centrifugal Pump Performance" proceedings of The Fourth International Pump Symposium, 19-25.
[6] A. J. Patel., B, B. Patel., (2014), "Design and Flow through CFD Analysis of Enclosed Impeller" International Journal of Engineering Research & Technology, Vol.3, 1366-1373.
[7] S. Kumar., S.K.Mohapatra., B.K.Gandhi., (2013), "Modelling and simulation of flow distribution of centrifugal slurry pump" http//www.researchgate.net/publication/269120973 ,1254-1260.
[8] S.Rajendran., K.Purushothaman., (2012), "Analysis Of Centrifugal Pump Impeller Using ANSYS-CFX" International Journal Of Engineering And Technology, Vol.1, 1-6.
[9] R. Teggin., S. Hebbal., M.S.Hebbal., (2016), "Effect of Blade Geometry on Hydraulic Performance of the Centrifugal pump" International Journal of Innovation Research in Science, Engineering and Technology, Vol. 5, 12380-12387.
[10] B. E. Abulnaga., "Slurry System Handbook"McGraw-Hill.