International Journal of Scientific Engineering and Applied Science (IJSEAS) - Volume-1, Issue-5, August 2015
ISSN: 2395-3470
www.ijseas.com

CFD Approach in the Design of Radial Flow Centrifugal Pump Impeller

Haridass Ramasamy[1], Prabhu Prakash[2]
[1] Karpagam College of Engineering, Coimbatore, India
[2] DAIJO Technologies, Coimbatore, India

Abstract

Pumps are used in the process of transferring fluids from one place to other and these pumps have a vital role in the domestic and industrial areas. This project deals with the application and need of CFD analysis in the pump industries. For this purpose, we have made a design and analysis of impeller used in Domestic Open well Radial flow water pumps. The impeller selected is of enclosed type, which is commonly used in domestic water pumps. In this project we have designed an impeller for a domestic need using formulas formulated by *Dr. K.M Srinivasan*, in the book *Rotodynamic Pumps*. The impeller is modelled using CAD software and analyzed using CFD package. The CFD output is cross checked with desired requirements, so as to state the accuracy and need of CFD analysis.

Keywords: *CFD analysis, Open well pump, Radial flow impeller.*

1. Introduction

Pump plays a vital role in transfer of fluids, as they are the basic source of suction. These pumps are of many types, based on their construction, type of operations, flow types, applications, etc., Design of such pumps are made using the thumb rule which is practiced in the industries. Standard design procedure have not been yet defined, as there are many design formulas formulated by great scientists like *Labanoff [1], John Tuzson, Stepanoff, KM Srinivasan [2], Turton*, etc., Since these formulas are theoretical, validation of the final product is obtained after many trials leading to Man, Machine and Money wastages.

To avoid this type of wastages, scientists have designed many software which are capable of analyzing the various flow parameters under varied flow conditions. The hydraulic part of the pump comprises of Casing and Impeller. In this work, the design of Impeller is carried out, whereas the casing design is dependent of impeller design.

2. Impeller

The impeller is a rotating component of a centrifugal pump, usually made of iron, steel, bronze, brass, Aluminum or plastic, which transfers energy from the motor that drives the pump to the fluid being pumped by accelerating the fluid outwards from the center of
Aluminum or plastic, which transfers energy from the motor that drives the pump to the fluid being pumped by accelerating the fluid outwards from the center of rotation. The velocity achieved by the impeller transfers into pressure when the outward movement of the fluid is confined by the pump casing. Impellers are usually short cylinders with an open inlet (called an eye) to accept incoming fluid, vanes to push the fluid radially, and a splined, keyed or threaded bore to accept a drive-shaft.
Impellers can be Open, Semi-open, or Enclosed. The open impeller consists only of blades attached to a hub. The semi-open impeller is constructed with a circular plate (the web) attached to one side of the blades. The enclosed impeller has circular plates attached to both sides of the blades. Enclosed impellers are also referred to as shrouded impellers.

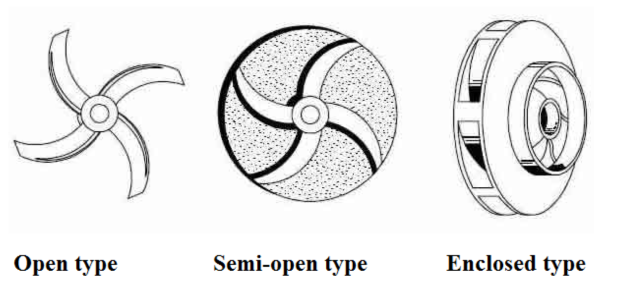

Fig. 1 Types of Impeller

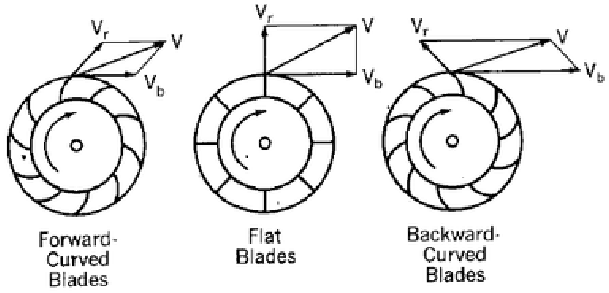

Fig. 2 Types of Vane profile

3. Impeller Design

Design is the application of scientific and mathematical principles to practical ends to form efficient and economical structures, machines, processes, and systems. Design of centrifugal impeller is done using K. M. Srinivasan method. Impeller design parameters are calculated using his procedure by giving head, volume flow rate and pump speed as input.

3.1 Specification of pump

Head: 24m
Discharge: 95lpm = 1.583lps
Power: 1hp = 746W
Speed: 2880rpm
Pipe size: 25 X 25 mm

3.2 Calculated parameters

The following are the parameters calculated using the above said method and these are the parameters which help in generating the impeller vane profile.

Specific Speed = 38.5
Power input to the pump = 1.84hp
Shaft Diameter = 25 mm
Outer Diameter of impeller = 144 mm
Velocity of fluid at the impeller inlet = 5.43 m/s
Inner Diameter of impeller = 36 mm
Inlet blade angle = 19.25^o
Impeller width at inlet = 10 mm
Blade angle at outlet = 23.76^o
Number of blades = 4

3.3 Vane profile development

The vane profile can be developed by using Point by Point method, Single arc method, Multi arc method and Error Triangle method. In this work, Point by Point method is selected, and the vane profile parameters are calculated and the profile is traced. The more number of points we employ, the tracing of the profile is made easy. In this case we have selected 7 trace points.

Table 1: Vane Profile Parameters

S. No	r	Cm	b	w	C_m/w	δ	t
1	18	1.44	32	5.46	0.26	5	28.27
2	27	1.39	28.3	5.06	0.27	5	42.41
3	36	1.34	24.7	4.67	0.29	5	56.54
4	45	1.29	21	4.27	0.30	5	70.68
5	54	1.25	17.3	3.87	0.32	5	84.82
6	63	1.2	13.7	3.48	0.35	5	98.96
7	72	1.15	10	3.08	0.37	5	113.09

Table 2: Vane Profile Parameters (continued)

δ/t	sin ß	ß	B	Δr	x
0.17	0.44	26.14	113.18	0.009	99.94
0.11	0.39	23.13	86.69	0.009	77.52
0.08	0.37	22.11	68.35	0.009	61.70
0.07	0.37	21.98	55.05	0.009	49.98
0.05	0.38	22.40	44.91	0.009	40.87
0.05	0.39	23.30	36.83	0.009	33.50
0.04	0.41	24.72	30.16	-	-

Table 3: Vane Profile Parameters (continued)

$\Delta\theta$	θ	
	Radians	Degrees
0.89	0	0
0.69	0.895	51.5
0.55	1.597	91.5
0.44	2.156	123.3
0.36	2.602	149.1
0.30	2.970	170.2
-	3.272	187.5

From the above table, the coordinate points of “r” and “θ” are taken and the vane profile is traced. The following image represents the vane profile traced using Uni Graphics software and this impeller is with forward curved vanes.

International Journal of Scientific Engineering and Applied Science (IJSEAS) - Volume-1, Issue-5, August 2015
ISSN: 2395-3470
www.ijseas.com

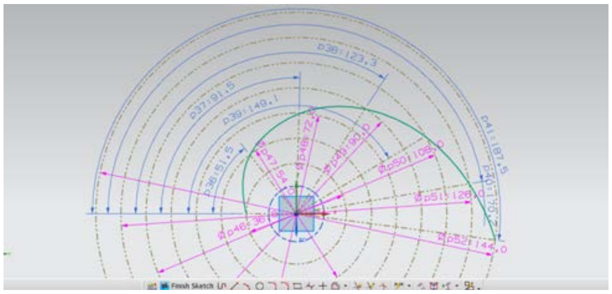

Fig. 3 Vane profile traced using CAD package

3.4 Modelling using CAD software

The above profile is then developed into a model using Uni Graphics. Some considerations are made while developing the model. Considerations like, the impeller is without hub portion, the shrouds are flat, etc., this made because, in this work, we have concentrated in the analysis of impeller's vane profile. Hence the parameters associated with Casing are neglected. The following image is the model developed using Uni Graphics.

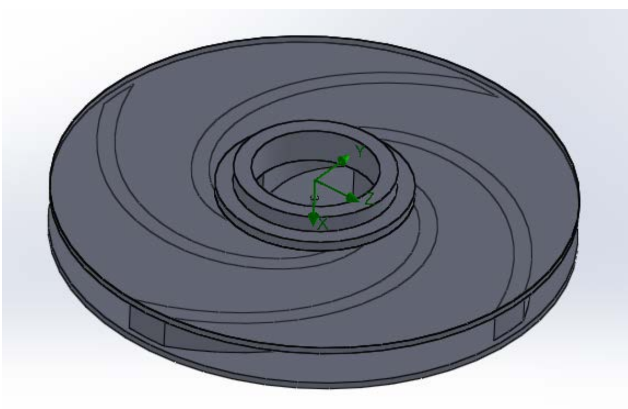

Fig. 4 Impeller modeling using CAD package

4. CFD analysis

The designed impeller is analyzed using a CFD package. For our work, we have used Fluid Flow Simulation which is a product of Solid Works. The following are the assumptions made while using CFD approach.

1. Incompressible flow
2. No-slip boundary condition
3. Gravity effects are negligible
4. Fluid properties are not functions of temperature

4.1 Boundary Conditions

Boundary conditions plays a major role in any mode of analysis, because, the output of any analysis is purely based on the input parameters. The following image shows the boundary conditions like Inlet velocity, Environmental pressure and rear wall. In this analysis, a circular ring is modelled around the impeller vane outlet, this is to capture the outlet flow parameters. It acts like a dummy wall.

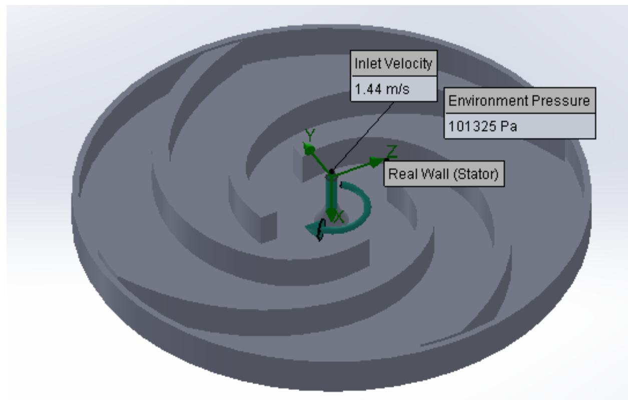

Fig. 5 Boundary Conditions used for CFD analysis

4.2 CFD results

After setting up the input parameters, the solver is set to compute the parameters, and the flow properties are traced as contours and vectors.

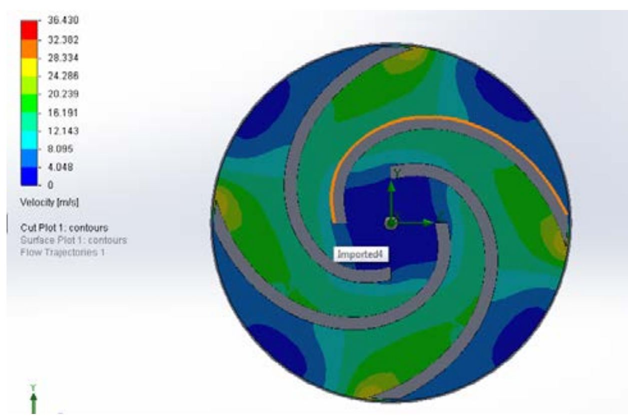

Fig. 6 Velocity plots of water inside the Impeller

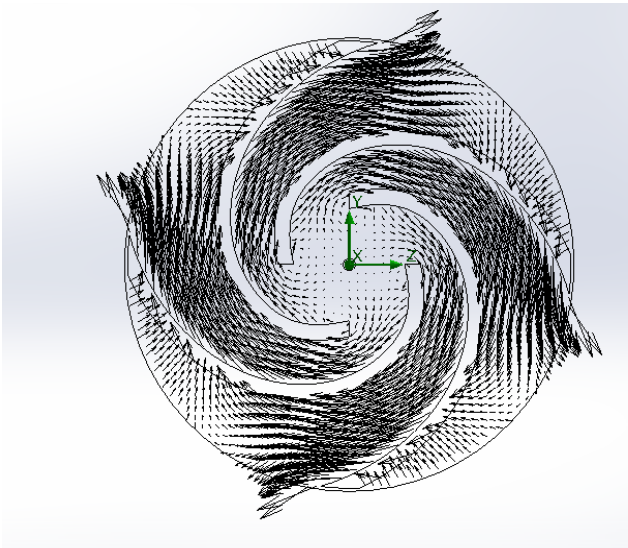

Fig. 7 Velocity Vectors of the water flow

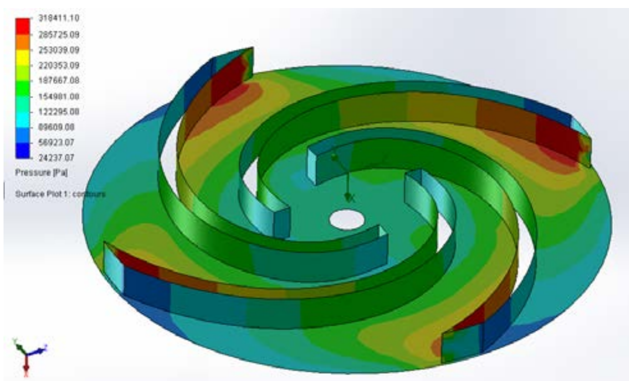

Fig. 8 Pressure plots of the water inside Impeller

Total pressure developed in the fluid (P) = 141744.4773 Pa

Fluid flow rate (Q) = 0.001445 m3/s

Total moment created by water on the impeller (T) = 10.4104 Nm

Hydraulic efficiency of the impeller $\eta h = \frac{PQ}{\omega T}$

Hh = 65.5 %

Total head generated (H) = $\frac{P}{\rho g}$ = 30.62

The discharge obtained by CFD simulation is 0.00144 m3/s whereas expected discharge is 0.00153 m3/s. Reduction in discharge occurs due to the losses (Friction, Turbulence) that occurs in impeller.

4. Conclusion

An impeller was designed for the input details using standard formulas and the vane profile was traced accordingly. Since the simulation is done only for the impeller's hydraulic part, the front and rear shrouds and hub portions are neglected in the modelling. The output of the simulation is close enough to the theoretical calculations. Hence it can be stated that, the usage of CFD analysis is worthy when compared to trial and error methods, thus reducing the time for various prototypes, and reducing the financial investments for each trials.

4.1 Future works

The future works for this paper will be designing an appropriate casing for the above designed impeller and conducting a combined flow analysis to predict the actual performance of the pump.

References

[1] Val S. Lobanoff and Robert R. Ross, Centrifugal Pumps, Design & Application, Second Edition: Woburn Ma, USA, Gulf Publishing Company, 1992.

[2] Rotardynamic Pumps: Centrifugal And Axial, K.M Srinivasan, First Edition, New Delhi, India, New Age International (P) Ltd., Publishers, 2008.

[3] R. Badie and J.B. Jonker, Finite Element Calculations and Experimental Verification of the Unsteady Potential Flow In A Centrifugal Volute Pump, International Journal for Numerical Methods in Fluids, Vol. 19, 1994, 1083-1102.

[4] Pham Ngoc Son, Jaewon Kim And E. Y. Ahn, Effects Of Bell Mouth Geometries On The Flow Rate Of Centrifugal Blowers, Journal Of Mechanical Science And Technology, Vol. 9, 2011, 2267~2276.

[5] R. Spence and J. Amaral-Teixeira, a CFD Parametric Study of Geometrical Variations on the Pressure Pulsations
And Performance Characteristics of a Centrifugal Pump, Elsevier, Computers & Fluids, Vol. 38, 2009, 1243–1257.

[6] L.E Reece, Some Fundamental Aspects of Centrifugal Pump Design in 11th Australian Fluid Mechanics Conference, 1992.