DESIGN AND ANALYSIS OF CENTRIFUGAL PUMP IMPELLER USING ANSYS FLUENT

Ajith M S[1], Dr Jeoju M Issac[2]

Abstract— **In this study, Computational Fluid Dynamics (CFD) approach was suggested to investigate the flow in the centrifugal pump impeller using the Ansys Fluent. Impeller is designed for the head (H) 70 m; discharge (Q) 80 L/sec; and speed (N) 1400 rpm. Impeller vane profile was generated by circular arc method and point by point method and CFD analysis was performed for the impeller vane profile. Further the impeller was analyzed for both forward and backward curved vane. The simulation on vane profile was solved by Nevier-Stokes equations with modified k - ω turbulence model. The impeller-Velocity and pressure distribution were analyzed for these Impellers.**

Index Terms— **CFD, Semi Circular pipe, Solar flat plate collector, Operating conditions**

I INTRODUCTION

Centrifugal pumps are a sub-class of dynamic axisymmetric work-absorbing turbomachinery. Centrifugal pumps are used to transport fluids by the conversion of rotational kinetic energy to the hydrodynamic energy of the fluid flow. The rotational energy typically comes from an engine or electric motor. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward into a diffuser or volute chamber (casing), from where it exits. Centrifugal pumps are widely used in many applications, so the pump system may be required to operate over a wide flow range in different applications. The most previous numerical studies were focused on the design or near-design state of pumps. [1]

Common uses include air, water, sewage, petroleum and petrochemical pumping. The reverse function of the centrifugal pump is a water turbine converting potential energy of water pressure into mechanical rotational energy. It is essential for a pump manufactured at low cost and consuming less power with high efficiency.[2] The overall performance is based on the impeller parameters and it is essential to identify the optimized design parameter of the impeller. CFD helps the designer to identify the optimal parameters of the impeller by numerical flow simulation.[3-5] Applying classical mechanics theory, assuming viscosity of the liquid equal to zero and no energy loss for the work of energy transferring from impeller to the streamlines which means that, all separate flow will be uniforms (this approximations of physical reality to get the simpler as solid state mechanism than hydraulic mechanism)[6-7]

Radial flow centrifugal pumps are widely used where head and discharge required are moderate. In radial type vanes, the vane profile is a curve that connects the inlet and outlet diameter of the impeller. Infinite number of curves can be drawn between two points, the length of the vane and hence the passage length can be different for same diameter D1 and D2 and same blade angle β1 and β2. Hence, it is necessary to define the shape of the vanes.[8] If the length of the passage is short, the divergence angle may increase gradually which will result in separation of flow and formation of eddies. In a longer passage frictional loss will be more. Only an appropriate passage length will give minimum losses. For designing an efficient blade profile an appropriate blade design method should be selected [9-10]

General methods available to design radial flow impeller vanes are simple arc method, double arc method, concentric circular arc method and point by point method. Here concentric circular arc method and point by point method methods are discussed in detail along with the calculations of blade design Further modeling of pump impeller with these methods and CFD analysis can be carried out to obtain the performance curve for comparing the efficiency and head obtained with different blade design method at various discharge conditions [11].

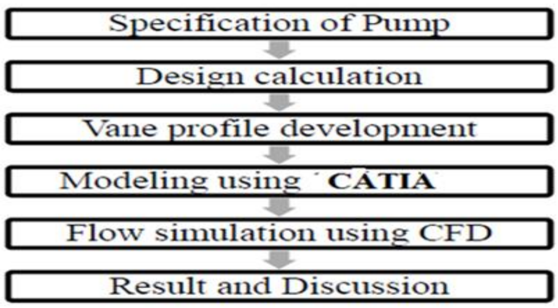

Figure 1 methodology

Ajith M S, *PG Scholar, Department of Mechanical Engg, Mar Athanasius College of Engineering, Kothamangalam 9497277803*

Dr Jeoju M Issac, *Professor, Department of Mechanical Engg, Mar Athanasius College of Engineering, Kothamangalam*

II METHODOLOGY

2.1.1 specification of the pump

A radial pump specification from the standard data is selected for design and analysis. Pump specification are

Head =70m
Discharge=80litres/s.
Rpm=1400

The impeller was designed for the operational condition of head (H) = 70m; flow rate (Q) = 80litre/sec; and speed (N) = 1400rpm

2.1.2predesign specific speed

Pre design specific speed is calculated using the empirical relation shown and nq=16.8 and it falls in the radial pump range.

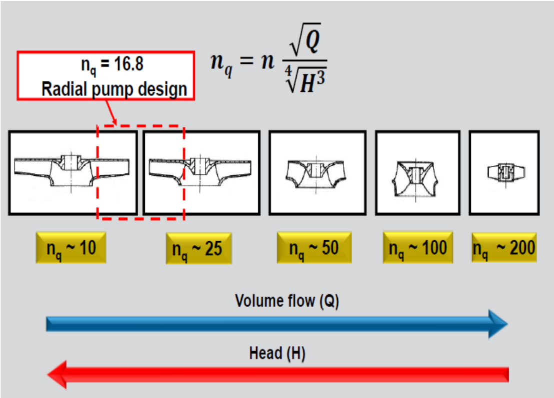

Figure 2.1 specific speed

2.2design calculations

2.2.1 meridion design

By using Cordier diagram and value of specific speed (б) the optimum values of outer diameter (D2) shroud diameter (Ds) and width of trailing edge are calculated. These values strongly decides the capitation behavior of the pump

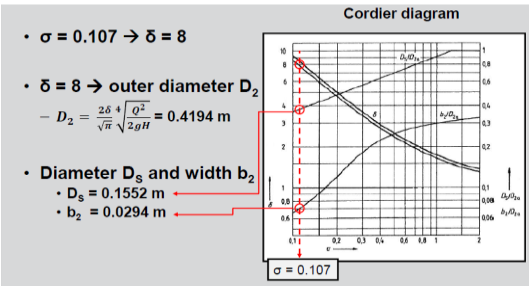

Figure 2.2 cordier diagram

The remaining dimensions are selected accordingly from the hand book to reduce the chance of cavitations. Inlet diameter is slightly larger than shroud diameter and leading edge is slightly larger than the trailing edge .

Inlet diameter (D1)=0.172m
Leading edge width (b1) =0.05m

2.2.2 blade ange design

By using inlet and outlet velocity triangles the inlet and outlet blade angles are calculated .Inlet velocity triangle is drawn flow is assumed to be radial at inlet and meridian component of velocity is calculated in such way that it is slightly higher than velocity at impeller eye

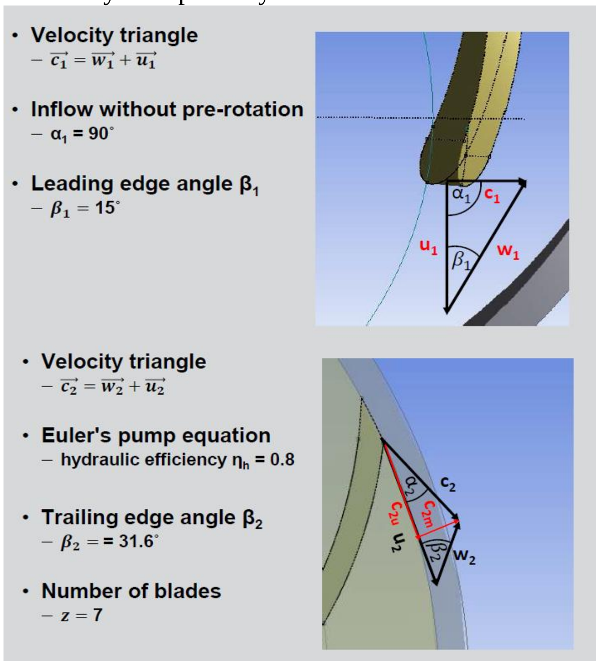

Figure 2.3 Velocity triangles at inlet and outlet

2.3.vane profile development

Vane profile is developed using two methods

1. **Circular arc method**
2. **Point by point method**

2.3.1 circular arc method

The impeller is arbitrarily divided into a number of concentric rings between r1 and r2. The radius of arc ρ, between any two rings rb and ra is obtained from the Equation. The computations of the values of ρ for various rings and the radii of arc of the vane are shown in Figure 5

$$\rho = \frac{rb^2 - ra^2}{2(rbcos\beta b - racos\beta a)} \quad (1)$$

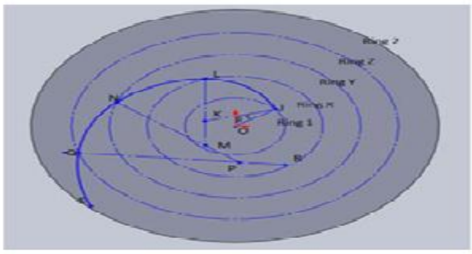

Figure 2.4 **Circular arc method**

2.3.2 point by point method

The co-ordinates for developing the vane profile together with the inlet and outlet angle depends on the radius (r). Tabular integration method is used for obtaining the co-ordinates. The radiuses with respect to angle are obtained from the Equation . The values of the vane profile coordinates and the vane profile are shown in figure6

$$r\partial\theta = \frac{\partial r}{tan\beta} \qquad (2)$$

$$\theta = \frac{180}{r}\int_{rb}^{ra} \frac{\partial r}{rtan\beta} \qquad (3)$$

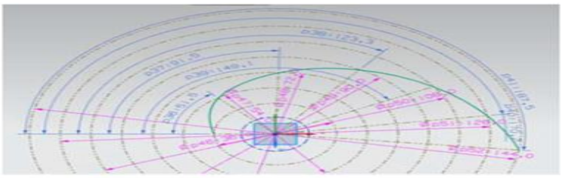

Figure 6 Point by point method

2.4modeling and meshing

Geometry of point by point impeller and Circular arc impeller are created using Catia V5 software. The grids are generated using ICEM CFD. The basic principle in meshing is that it should have finer elements to get better accuracy of the result. At the same time, number of grids should not exceed available computational capacity. To achieve this objective, fine mesh was used where the solution gradient is higher and coarse mesh was used where there is low solution gradient.

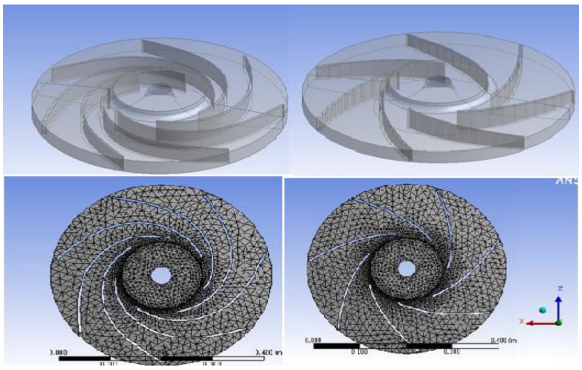

Figure 2.5 Geometry and meshed models

2.5 flow simulation using cfd

CFD approach was carried out to analyze the behavior of flow field in the impeller using the Ansys fluent 14.5 software. Fluent software is a powerful CFD tool that enables designers to quickly and easily simulate fluid flow for the success of designs. Design cycles are expensive and time-consuming. CFD analysis is able to help the designers to optimize the designs by simulating several concepts and scenarios to make absolute assessment. Fluent solves time-dependent three-dimensional Reynolds-averaged Navier-Stokes equations using the k-omega turbulence model with the Finite Volume Method (FVM)

2.5.1 governing equations

The principles of conservation law governed by fluid dynamics are

Mass continuity:

$$\frac{\partial\rho}{\partial t} + \nabla.(\rho u) = 0 \qquad (4)$$

Navier stoke's:

$$\rho\left(\frac{\partial u}{\partial t} + u.\nabla u\right) = -\nabla p + \nabla T + f \qquad (5)$$

Energy :

$$\rho\frac{DE}{Dt} = -\rho\nabla u + \nabla.(k\nabla T) + \emptyset \qquad (6)$$

2.5.2 k-ω turbulence model

A two transport equation model solving for *k* and ω the specific dissipation rate (ε /*k*) based on Wilcox. This is the default *k*-ω model. Demonstrates superior performance for wall bounded and low-Re flows. Shows potential for predicting transition.SST K-ω A variant of the standard-ω model, Combines the original Wilcox model (1988) for use near walls and standard k-ε model away from walls using a blending function. Also limits turbulent viscosity to guarantee that $\tau_{t,} K$.

2.5.3 boundary conditions

The operating pressure was taken as 0 Pa, since all the experimental measurements are taken at absolute pressure. Inlet velocity is defined as boundary condition at inlet and at outlet mass flow rate is defined as boundary condition, and Relative velocity of the impeller is set as global rotating reference frame.

2.6 results and discussions.

Four impellers CAFC, CABC, PPFC and PPBC were analyzed using Ansys fluent 14.5. The analyses were made for the circular arc method and point by point method with forward and backward curved vanes. The results of the flow field investigation are presented in terms of velocity and pressure distribution of the impeller passages. Figure7 shows the velocity distributions and pressure distributions on the vane-to-vane for impeller CAFC, CABC, PPFC and PPBC, respectively. Noting the fact that flow distributions in the backward curved vane have high efficiency compared with forward curved vane. Hence it is evident from Figure the backward curved vanes have better flow distribution than the forward curved vane. The pressure increases normally on the pressure Surface than on the suction surface on each plane. Impeller CABC has gradual pressure distribution in the stream wise direction than other impellers as shown in Figure. The maximum efficiency of the impeller was obtained for the backward curved vane profile. However, the maximum efficiency is 58.53% for the backward curved circular arc method

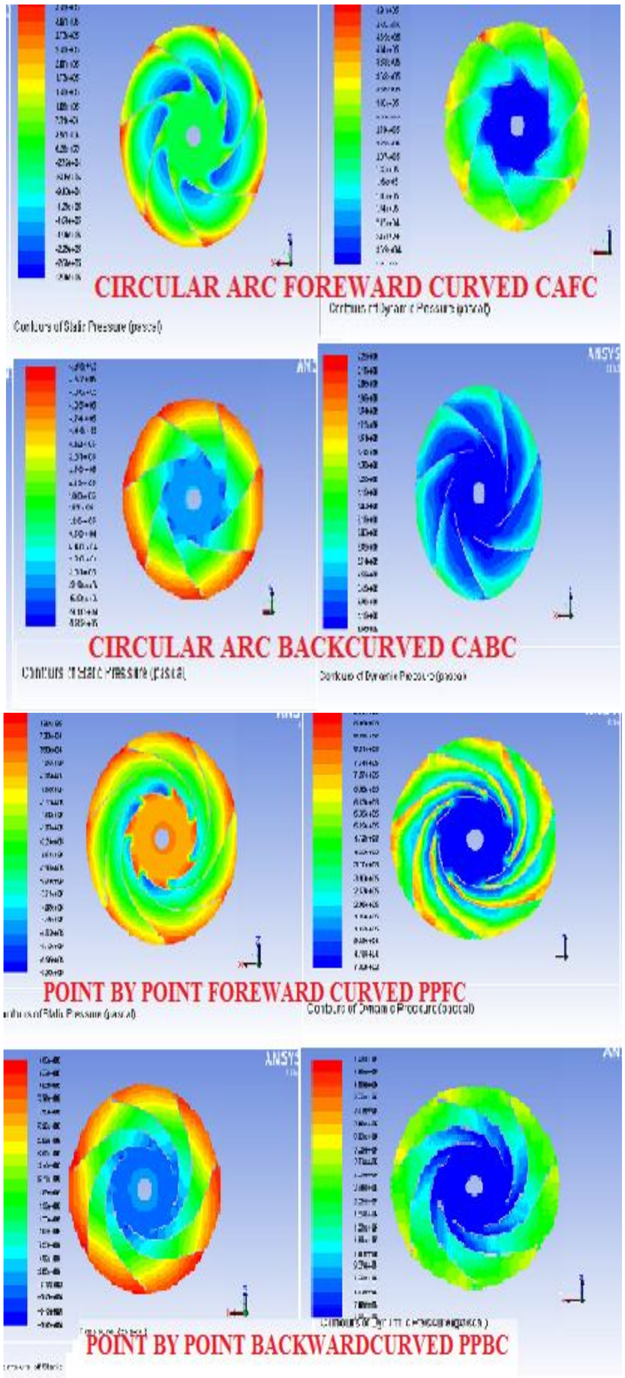

Figure 2.6Pressure and velocity distributions in PPFC PPBC CAFC AND CABC.

III CONCLUSIONS

Numerical investigations were carried out to analyze the flow field in the pump impeller using Ansys fluent 14.5. To design a centrifugal pump impeller a procedure is proposed. The design procedure leads to good results in a lesser time. The effect of the forward curved vane and backward curved vane were analyzed. From the numerical results the backward curved vanes have better performance than the forward curved vane. The vane profile was developed by two methods viz. circular arc method and point by point method. The flow simulation gives a result that both of the methods i.e. circular arc and point by point leads to similar head rise. Therefore whichever is easier to manufacture that can be followed by designer.

ACKNOWLEDGMENT

I have great pleasure in submitting this paper. It give me immense pleasure to record my debt of gratitude and my warmest regards to all the faculties of Mechanical Department. The various values that tried to learn from them shall remain a source of inspiration for me forever.

I am thankful to my family and friends for their whole hearted blessings are always for me support and constant encouragement towards the fulfillment of the work

REFERENCES

[1] *S R Shaha et al.* CFD for centrifugal pumps stae of the art, Chemical, Civil and Mechanical engineeringTracks of 3rd Nirma University International conference journal

[2] *Peter Hlbocana et al.* Prime Geometry Solution of a Centrifugal Impeller Within a 3D Setting, XIIIth International Scientific and Engineering Conference "HERVICON-2011"

[3] *Raghavendra S Muttalli et al.* CFD Simulation of Centrifugal Pump Impeller Using ANSYS-CFX, International Journal of Innovative Research in Science, Engineering and technology Vol. 3, Issue 8, August 2014

[4] *Tilahun Nigussie et al*. Design and CFD Analysis of Centrifugal Pump, International Journal of Engineering Research and General Science Volume 3, Issue 3, May-June, 2015 ISSN 2091-2730

[5] . *Maitelli et al.* Simulation of flow in a centrifugal pump of esp systems using computational fluid dynamics, BRAZILIAN JOURNAL OF PETROLEUM AND GAS , v. 4 n. 1 ,p. 001-009 ,2010 ,ISSN 1982-0593

[6] *Mehul et al.* CFD analysis of mixed flow pump impeller, International Journal of Advanced Engineering Research and Studies,E-ISSN2249–8974

[7] *Alpeshkumar et al.* CFD Analysis and Experimental Study on Impeller of Centrifugal Pump, *IJSRD - International Journal for Scientific Research & Development| Vol. 3, Issue 02, 2015 | ISSN (online): 2321-0613*

[8] *S.Rajendran et al.* Analysis of a centrifugal pump impeller using ANSYS-CFX, International Journal of Engineering Research & Technology (IJERT)Vol. 1 Issue 3, May - 2012ISSN: 2278-018

[9] *P.Gurupranesh et al.* CFD Analysis of centrifugal pump impeller for performance enhancement, *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X PP 33-41*

[11] *Ashish Bowade et al* A Review of Different Blade Design Methods for Radial Flow Centrifugal Pump,International Journal of Scientific Engineering and Research (IJSER) www.ijser.in ISSN (Online): 2347-3878

***Ajith M S**, PG Scholar, Department of Mechanical Engg, Mar Athanasius College of Engineering, Kothamangalam 9497277803.*

***Dr Jeoju M Issac**, Professor, Department of Mechanical Engg, Mar Athanasius College of Engineering, Kothamangalam*

