ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 4, Special Issue 16, April 2017

Design of Impeller and Analysis of Flow in Centrifugal Pump

Karan Kumar Jalan O.[1], Anand A.S.[1], Dr. V. Santhanam[2],
[1]Student,UG, Department of Mechanical Engineering, Rajalakshmi Engineering College, Chennai.
[2]Professor, Department of Mechanical Engineering, Rajalakshmi Engineering College, Chennai.

Abstract **- An impeller is a radial type rotor that increases the acceleration of the liquid due to the centrifugal action and the liquid gets pumped through the casing to a higher head. There are certain design parameters for centrifugal pump that can be optimized, such as to improve the efficiency and flow rate of the liquid. Based on the theory of pump characteristics, the impeller design and development of potential flow calculations for the liquid flow using the MATLAB software was performed. NX-CAD was used to model the impeller and pump. CFD analysis in ANSYS CFX showed the flow of liquid through the variable cross-section of the volute casing of the pump, virtually. The simulation of the liquid flow based on the design determined the errorless performance. Modification of the impeller design resulted in the enhancement of the overall performance of the pump based on efficiency, power and potential flow rate of the liquid. The program developed in the MATLAB also served as a type of calculator for determining the type of pump that can be used for various industrial and household purposes based on the velocity of the liquid, head and discharge rate.**

Keywords**-Impeller, Centrifugal pump, MATLAB, NX-CAD,ANSYS CFX**

I. INTRODUCTION

The Impeller of the Centrifugal pump usually has vanes fitted between the shrouds or plates. The crown plate has a suction eyes and the base plate is mounted which is keyed to a shaft. The impeller without the crown plate is known as the non-clog or semi-open impeller. Ajith M S et al [1] have suggested the use of CFD approach to investigate the flow in the centrifugal pump impeller using Ansys Fluent.Numerical investigations were carried out to analyze the flow field in the pump impeller using Ansys fluent. The velocity of the impeller and pressure distribution were also analyzed for the Impellers.

The overall performance of the centrifugal pump is based on impeller parameters and it is necessary to identify the optimized design parameter of the impeller. Kulkarni et al [2] proposed a Parametric Study of Centrifugal Pump and its Performance Analysis using CFD. The design and performance analysis of the pump components were done and the efficiency was found out which was satisfactory.

CFD helps in identifying various optimal parameters of impeller by executing various numerical flow simulations in this project work we are using radial flow centrifugal pump which is widely used where head and discharge required are moderate. The vane profile is a curve that connects the inlet and outlet diameter of the impeller. E.C. Bacharoudis et al [3] worked on the Parametric study of a centrifugal pump impeller by varying the outlet blade angle.The numerical simulation seem to predict reasonably the total performance and the global characteristics of the laboratory pump as a final outcome. The program that was developed in the MATLAB served as a calculator to determine the type of pump that can be used for various industrial and household purposes based on the velocity of the liquid, head and discharge rate. Dr. Mohammed Ali Hussain et al [4] has done pioneering work to establish design methodology for pumps, fans and blowers.It was aimed to simulate the three dimensional complex internal flow in a centrifugal pump impeller with a five twisted blades by using CFD. As a result Flow indicates appropriate constant velocity vectors. M.G.Patel, et al [5] worked on Effect of impeller blade exit angle on the performance of centrifugal pump published.To investigate effect of blade exit angle on performance of the centrifugal pump.

Design of Impeller

There are two types of impellers, depending on the flow regime created.

- Axial flow impeller
- Radial flow impeller

Impellers consist of various vanes often blade shaped arranged around a short central shaft. When the shaft and vanes rotate, they suck in fluids or gases and impel them out the other side. Pump impellers rely on Bernoulli's principle which states that an increase in the liquid velocity occurs along with the decrease in pressure or potential energy, and vice versa, to operate.

When the liquid enters the pump, it becomes trapped between the vanes of the impeller and the pump casing wall and increases in velocity as it moves from the impeller eye (Centre) toward the outer diameter of the impeller. Once the liquid reaches a certain point near to the outside diameter, it abruptly decreases in velocity and experiences an equal increase in pressure. The liquid

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 4, Special Issue 16, April 2017

becomes even more pressurized as it is been discharged from the impeller and out of the pump orifice. Based on the principles of operation of the pump, it can be understood that the impeller of the pump's rotational speed and vane height (affects pressure and the possible formation of a vacuum) determines the pump's output pressure and flow.

II. DESIGN CALCULATIONS

To compute the various parameters of the pump, Equations 1 – 10 were used as follows [6]

- Specific speed is a term used to describe the geometry (shape) of a pump impeller.

  Specific speed (N_s) = √— / —— (1)

- The term Nominal Diameter refers to the internal diameter of a pipe

  Nominal Diameter (D_1) = √— (2)

- Output Power (P_o) :
  Output power assumed as 1 hp

- Input Power to Motor (P_1) = —hp (3)

- Volumetric Efficiency the ratio of the volume of fluid actually displaced by a piston or plunger to its swept volume

  (η_v) = ————% (4)

- Mechanical efficiency measures the effectiveness of a machine in transforming the energy and power that is input to the device into an output force and movement.
  Mechanical Efficiency (η_m) = Assume η to be as 86%

- Overall efficiency looks at entire systems from the initial input to the final output. Overall Efficiency (η_o) = η_v x η_m % (5)

- Shaft diameter is the diameter of the hole in the iron laminations that contains the shaft.

  Shaft Diameter (d_s) = √—— (6)

- Velocity inlet boundary conditions are used to define the flow velocity, along with all relevant scalar properties of the flow, at flow inlets

  Inlet Velocity (U_1) =———m/s (7)

- The blade angle $\beta_b(r)$is defined as the inclination of the tangent to the blade in the median plane and the plane perpendicular to the axis of rotation.
  - Inlet Blade Angle : $\tan^{-1}$ (8)
- Out Blade Angle : $\tan^{-1}(V_{f2}/(u_2-V_{w2}))$ (9)
- Outlet Velocity (U_2) = ($*D_2*N$)/60 (10)

III. CAD MODELING OF THE IMPELLER

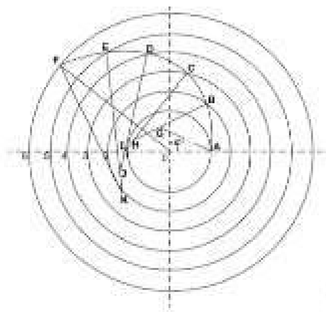

Fig. 1. Blade profile [7]

A. *Procedure for Impeller Design*

- The design procedure employs the basic laws of fluid motion to determine the required flow path.
- Initially, two concentric circles are drawn whose diameter equals the internal and external diameter of the impeller.
- In the next step, three to five concentric circles are drawn at equal distance starting from the internal diameter till the external diameter.
- Then, a line is drawn from point A at an angle of the inlet blade angle till the distance of first radius of curvature value.
- With G as center and AG is radius, an arc is drawn till the arc meets the next circle.
- A line is drawn from B, which passes through the previous center and the length is equal to the second radius of curvature.
- With H as center and BH is radius, an arc is drawn till the arc meets the next circle.
- This process is repeated till the curve meets the external diameter of the impeller.
- The curve is then off-set with internal blade thickness at the entry of the blade and external blade thickness at the exit of the blade given in below figure 2.

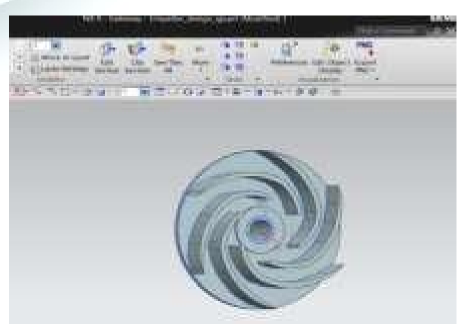

Fig. 2. NX-CAD design of impeller

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 4, Special Issue 16, April 2017

IV. MATLAB

It is considered as a programming language developed by Math Works in order to simplify the calculations. MATLAB allows matrix manipulations, data implementation on algorithms and plotting functions, interfacing and creation of user interfaces, with programs written in other languages such as C++,python. Any simulation, plotting, coding can be executed using MATLAB.

It is a tool for programming, graphs, simulations, graph, automation, measurement and statistics, from an engineer's point of view. MATLAB can be considered as essential coding language for engineers for doing calculations. It contains diverse set of mathematical commands which are simple to implement.

Therefore, while using MATLAB we can focus on the actual coding algorithms instead of trying to perform hard mathematical operation. For instance, to multiply an array of numbers with another array you need to run a loop, whereas, using MATLAB you simply write multiply command and MATLAB will do multiplication in element wise order to form a third array. This is a small example and the world of MATLAB commands is immensely huge and interesting.

A. *MATLAB Programming for best pump selection*

The pump performance curve is initially loaded into the Engauge digitizer software. Then, the data points are derived from the pump curve and exported to the excel sheet using .csv extension. This file is then loaded into the MATLAB program in the form of an array and the data is re-plotted again in the form of a graph. The program also calculates all the parameters of the pump using the formulae obtained from the velocity triangle and then links the data points obtained with the pump performance curve and the data obtained from the formulae. The final graph obtained is used to choose the pump with best efficiency, mass flow rate and pressure head.

B. *Graph Plotted*

Characteristic curve are essential to know for choosing and purchasing a pump. The Figure 4 shows the re-plot of the pump performance curve using MATLAB. The design spot on the diameter curve indicates the selection of the optimum kind of pump for a particular application with the best efficiency available. The Figure 5 shows the plot of speed from 2000 rpm to 2400 rpm in steps of 50 rpm increment versus discharge rate.

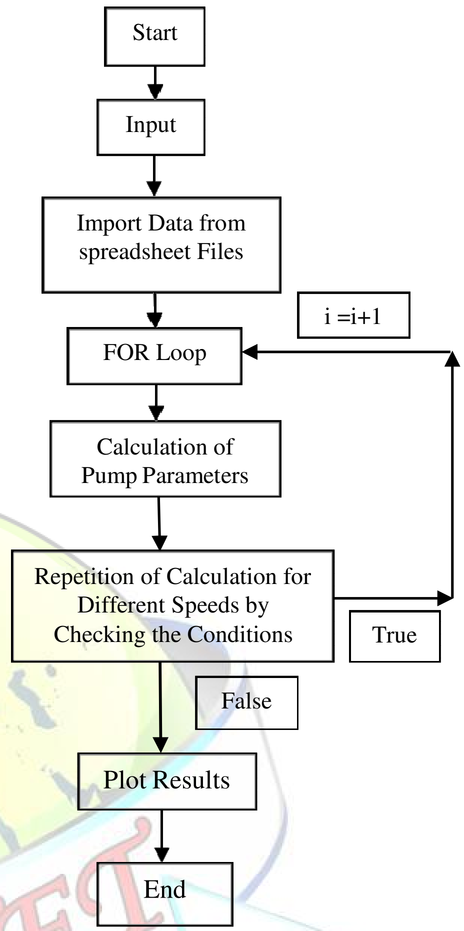

Fig.3. Flowchart for re-plot of the pump curve

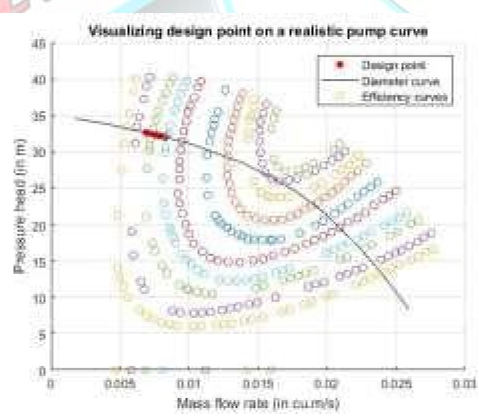

Fig.4. Re-plot of pump curve in MATLAB

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 4, Special Issue 16, April 2017

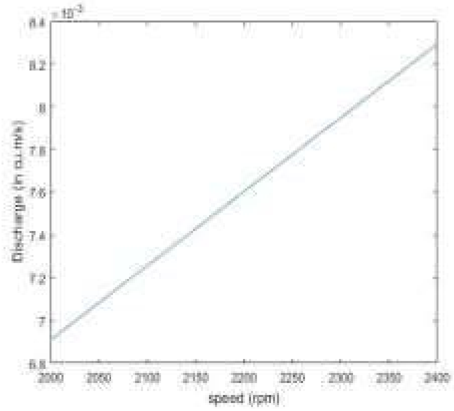

Fig. 5. Plot of speed versus discharge

V. CFD ANALYSIS OF THE FLUID FLOW IN THE PUMP

CFD can be defined or termed as a science of understanding and predicting fluid flow, mass transfer and related phenomena by solving the mathematical problems, chemical reactions and equations which govern processes using a numerical process. J H Kim et al [8] presented about Design optimization of a centrifugal pump impeller and volute using computational fluid dynamics. In this study, optimization of the impeller design of volute and optimization of the impeller were carried out in order to improve the performance of a centrifugal pump. Finally, a method for designing the volute was suggested via Stepanoff theory.

The CFD analysis result obtained is used in Conceptual studies of new designs, detailed product development and redesigning work. CFD analysis complements testing and experimentation, which are explained as follows.

- Analysis begins with mathematical model of a physical problem.
- Conservation of matter and energy must be satisfied throughout the region of interest.
- Simplifying assumptions are made in order to make the problem tractable.
- Provide appropriate initial boundary conditions for the problem.
- CFD includes numerical methods to develop discretized equations of the governing equations in fluid mechanics and heat transfer.
- The solution is post-processed to extract quantities of interest(e.g. lift, drag, heat transfer).

A. *Discretization*

- Domain is discretized into a finite set of control volumes. The discretized domain is called the mesh or grid.
- General conservation equations for mass, momentum, energy, etc. are discretized into algebraic equations.
- All equations are carried out to render flow field.

B. *Tools To Examine The Results*

- Graphical tools:
  - Grid, contour, and vector plots.
  - Particle trajectory plots.
  - Path line trajectory plots
  - XY plots.
  - Animations
- Numerical tools for reporting function
  - Flux balances.
  - Surface and volume integrals and averages.
  - Forces and moments

C. *Applications of CFD*

Applications of CFD are numerous.

- Flow and heat transfer in the industrial processes (boilers, heat exchangers, combustion equipment, pumps and blowers (piping manufacturing).
- Aerodynamics of ground vehicle and aircraft, missiles.
- Film coating, thermoforming in processing applications.
- Flow and heat transfer in propulsion and power generation systems.
- Ventilation, heating, and cooling flows in the building and industries.
- Chemical vapor deposition (CVD) for the integrated circuit manufacturing.
- Heat transfer for electronics packaging applications.

D. *Advantages of CFD*

- Speed
  - CFD simulations can be done in a short duration of time.
  - Engineering data can be introduced early in the design process.
- Capability to simulate real conditions.
  - Many flow and heat transfer processes cannot be experimented e.g. hypersonic flow condition. CFD provides the ability to simulate at any point of physical condition.
- Ability to create ideal conditions.
  - CFD allows a great control of the physical process, and provides ability to isolate specific phenomena for study process.
  - A heat transfer process can be controlled with adiabatic, constant heat flux condition or constant temperature boundary.
- Comprehensive information.

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 4, Special Issue 16, April 2017

 - Experiments only permit data to be extracted at a limited number of locations. CFD allows the analyst to examine a large number of locations in the region of interest, and yields a set of flow parameters for examination.

E. Limitations of CFD

- Physical models.
  - The solution obtained by CFD analysis, depends upon various physical models of real world processes (e.g. turbulence, compressibility, chemistry, multiphase flow, etc.)
  - The CFD solutions can only be as accurate as the physical models on which they are based.
- Numerical errors.
  - Round-off error which occurs due to finite word size available on the computer. Round-off errors will always exist though they are small in most cases.
  - Truncation error occurs due to approximations in the numerical models. Truncation errors will go to zero as the grid becomes refined. Mesh refinement is one of the ways to deal with truncation error.

F. Boundary Conditions Used In CFD Analysis

Boundary condition helps in solving the CFD problem with inlet and outlet conditions. The most common boundary conditions are Inlet conditions, Symmetry conditions, Physical boundary conditions, cyclic conditions, pressure conditions, exit conditions. In this problem, the boundary conditions that were used are rotational speed of the rotor, mass flow rate of the liquid inlet pressure of the liquid. The values of these above boundary conditions are mentioned in the table 1. The surface model that was used in the solving of the problem is shown in the figure 6. This figure also consists of the variable cross section of the casing.

Fig. 6. Impeller and casing design for CFD analysis

TABLE 1
BOUNDARY CONDITIONS USED IN CFD ANALYSIS

S.No.	Type of Boundary Condition	Values
1	Mass flow rate	0.35 m^3/kg
2	Inlet Pressure	1 bar
3	Rotation speed	2000 rpm
4	Number of Iterations	625

VI. ADVANTAGES

- The solution obtained can be used to solve complicated problems, improve efficiency and expand opportunities.
- It has a direct impact on identifying the required pump according to customer specification.
- CFD meets the goal better than any other theoretical or experimental methods followed
- This saves cost and turnaround time and it procures reliable results.

VII. APPLICATIONS

The program that has been developed in the MATLAB helps the industrialists in the ease of choosing the best type of pump that can be used, according to the requirement of the customer.

This also helps in reducing the time in choosing the pump.

VIII. RESULTS AND DISCUSSION

The design modification, which was done on the blades of the impeller showed an increase in the overall efficiency which was greater by 8% than the previous pumps. The programming in MATLAB helped in obtaining the best type of pump by the industrialists that can be delivered to the customer according to the latter's requirement.

While the CFD analysis showed the mass flow rate distribution along various cross section of the pump casing, while the velocity gradient distribution depicts the changes in the velocity of the fluid along the various cross section of the pump casing.

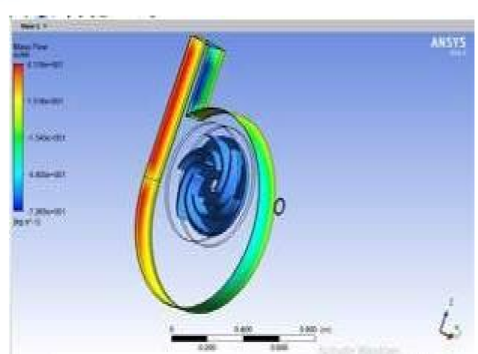

Fig.7. Mass flow rate distribution

ISSN 2394-3777 (Print)
ISSN 2394-3785 (Online)
Available online at www.ijartet.com

International Journal of Advanced Research Trends in Engineering and Technology (IJARTET)
Vol. 4, Special Issue 16, April 2017

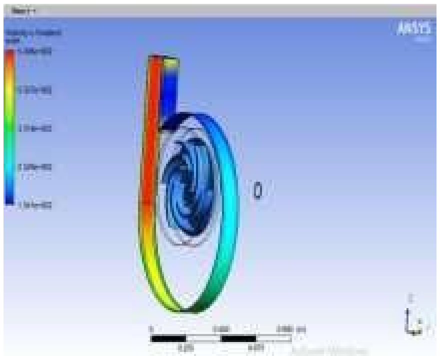

Fig.8. Velocity gradient of the liquid flow

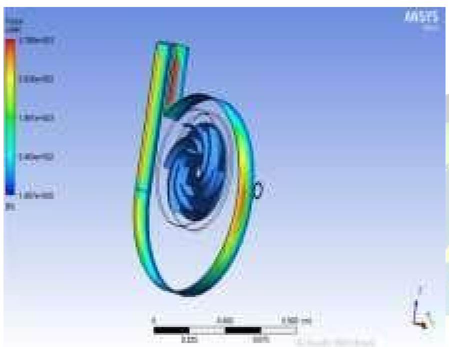

Fig.9. Force with which liquid exits the impeller

IX. CONCLUSION

Various numerical investigations were carried out to design an impeller and analyse the flow of liquid in the pump. The result obtained shows a promising better performance. The vane profile was developed in NX-CAD software by circular arc method. The MATLAB program helped in choosing the best suitable pump according to the requirement and the application. The CFD analysis showed the distribution of various parameters along various cross section of the pump which indicated the proper flow of the fluid.

A framework was built to perform simplified pump calculation and for displaying actual pump performance curve data for manufacturer.

A systematic procedure was developed to digitize experimental data from images to make use of in the program.

Linear interpolation was used to calculate realistic pressure drop data based upon the simplified pump model and manufacturer's model.

This framework can be applied to extract and store the performance model of 100s of pump model that can be used for commercial purpose.

Such a tool can be useful for designers and analyst in selecting the pump that meets the design and cost requirement.

ACKNOWLEDGEMENT

We have an immense pleasure and gratitude in submitting this paper. My warmest gratitude to all my staffs and faculties.

We are thankful to our friends and family for providing us their support which helped us move forward with this paper and encouragement towards the fulfillment of the work.

REFERENCES

[1] Ajith M.S. & Issac, J. M. design and analysis of centrifugal pump impeller using Ansys fluent.
[2] Kulkarni, S. S.(2014) Parametric Study of Centrifugal Pump and its Performance Analysis using CFD. International Journal of Emerging Technology and Advanced Engineering, 4, 155-161.
[3] Bacharoudis, E. C, Filios, A. E., Mentzos, M. D., and Margaris, D. P. (2008). Parametric study of a centrifugal pump impeller by varying the outlet blade angle. Open Mechanical Engineering Journal, 2(5), 75-83.
[4] Dr.Mohammed Ali Mahmood Hussain, Dr.Wajeeh Kamal Hasan, The effect of rotational speed variation on the velocity in the single blade passage centrifugal pump at IOSR JOURNAL OF MECHANICAL AND CIVIL ENGINEERING
[5] Patel, M. G., & Doshi, A. V. (2013). Effect of impeller blade exit angle on the performance of centrifugal pump. Int J. Emerging Technology and Advanced Engineering, 3(1), 91-99.
[6] Design and analysis of pump impeller using SWFS, ISSN1 746-7233, England, UK World Journal of Modeling and Simulation Vol. 10 (2014) No. 2, pp. 152-160
[7] Fluid mechanics by R K BANSAL
[8] Kim, J. H., Oh, K. T., Pyun, K. B., Kim, C. K., Choi, Y. S., & Yoon, J. Y. (2012). Design optimization of a centrifugal pump impeller and volute using computational fluid dynamics. In IOP Conference Series: Earth and Environmental Science (Vol. 15, No. 3, p. 032025).