Journal of Mechanical and Production Engineering (JMPE)
ISSN 2278-3512
Vol.2, Issue 2, Sep 2012 87-97

PARAMETRIC STUDY AND OPTIMIZATION OF CENTRIFUGAL PUMP IMPELLER BY VARYING THE DESIGN PARAMETER USING COMPUTATIONAL FLUID DYNAMICS: PART I

R.RAGOTH SINGH[1,2] & M.NATARAJ[3]

[1]Assistant Professor, Kathir college of Engineering, Coimbatore, India

[2]PhD Scholar, Anna University of Technology, Coimbatore, India

[3]Associate Professor, Government college of Technology, Coimbatore, India

ABSTRACT

In this study, the characteristics of low specific speed centrifugal water pump are presented. The characteristics are evaluated by studying the relationships among the impeller eye diameter, vane exit angle and width of the blade at exit. As these pumps are of Non-positive type, the discharge is greatly affected by any resistance to flow, outlet conditions and design parameters of impeller and casing. Therefore it is necessary to find out the design parameters and working conditions that yield optimal output and maximum efficiency with lowest power consumption. This paper is devoted to the performance evaluation of a centrifugal pump for the given specification. Different pump models are developed by varying critical design parameters to different levels. Response surface method is used for Experimental Design (DoE). Computational Fluid Dynamics (CFD) analysis is carried out on the developed models to predict the performance virtually and to verify with the experimental result of the pump. Optimal pump design is formulated using Response surface method. The objective functions are defined as the total head and the total efficiency at the design flow-rate.

KEY WORDS: Centrifugal Pump; Impeller; Doe; RSM; CFD.

INTRODUCTION

Centrifugal pumps are common for many different applications in the industrial and other sectors. Nevertheless, their design and performance prediction process is still a difficult task, mainly due to the great number of free geometric parameters involved. On the other hand the significant cost and time of the trial-and-error process by constructing and testing physical prototypes reduces the profit margins of the pump manufacturers. For this reason, CFD analysis is currently being used in hydrodynamic design for many different pump types [M.H. Shojaeefard et al, (2012), Jie Jin, Ying Fan, et al, (2012), R. Barrio, et al, (2012, E.C. Bacharoudis, et al, (2008), Sun-Sheng Yang, et al, (2012)]. The performance analysis of centrifugal pump is chosen because it is the most useful mechanical rotodynamic machine in fluid works which widely used in domestic, irrigation, industry, large plants and river water pumping system. These pumps are used at the places where the requirements of head and discharge are moderate. Many researches are going on in the field of centrifugal pump to improve the performance and to reduce the losses such as turbulence loss, shock losses, impeller friction losses, volute friction losses, disk friction losses and recirculation losses and also power consumption. Experimental investigations are generally carried out on pumps which are expensive, time consuming and limited to some extent. To reduce the number of experimental works, virtual analysis can be carried out on different pump models with the use of CFD software to predict pump performance.

A study was carried out with the aim of identifying the design parameters and modifying the design parameters to improve the pump performance characteristics [M.H. Shojaeefard et al, (2012), Jie Jin, Ying Fan, et al, (2012), R. Barrio, et al, (2012, E.C. Bacharoudis, et al, (2008), Sun-Sheng Yang, et al, (2012)]. This work is described in a two-part paper, of which this is the first part. In this paper (Part I), the primary design parameters are considered for the performance analysis numerically and experimentally, in which the design parameters were evaluated by CFD, was employed along with an intensive experimental study. This approach allowed the origin of the fluid working domain to be identified. A detailed discussion of the methodology [Barker Thomas B. (1985) , Kuehl Robert O (2000), Montogomery Douglas C.(2007)] as in "Figure 1" for optimizing the design parameters using Response Surface Method (RSM) can be found in Part II of the paper along with details of DoE carried out to improve the performance characteristics of the pump.

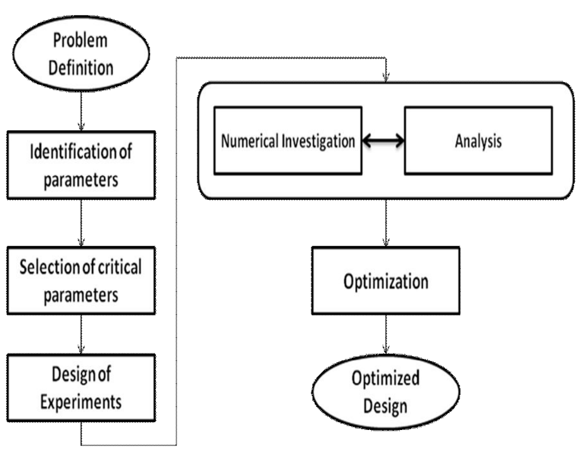

Figure.1. Methodology

LITERATURE SURVEY

A number of investigators have considered the effect of geometry modifications on the impeller and volute in centrifugal pumps have been carried out experimentally and numerically [B. Jafarzadeh, et al, (2012), Raúl Barrio, et al, (2010), John S and Anagnostopoulos, (2009) J. Fan, et al, (2011), Wang Ji-Feng, et al, (2012, H Chen, et al, (2010), R. Spence and J. Amaral-Teixeira, (2009) “A CFD parametric study of geometrical variations on the pressure pulsations and performance characteristics of a centrifugal pump,” An International journal: Computers & Fluids, 38, .1243–1257, R. Spence and J. Amaral-Teixeira, (2008) , Wei Hana, et al, (2012), Punit Singh and Franz Nestmann, (2010)]. The effects of blade outlet angle and passage width on the performance of a centrifugal pump have been investigated numerically and experimentally [M.H. Shojaeefard et al, (2012)]. Numerical simulation and performance tests were adopted to study the model of the centrifugal pumps, to analysis the hydraulic properties of the ultra-low specific-speed centrifugal pump [Jie Jin, Ying Fan, et al, (2012)]. The total radial loads on the impeller of centrifugal pumps under different operating condition were estimated by means of the numerical simulation of the unsteady flow with an appropriate CFD code [R. Barrio, et al, (2012)]. The influence of the outlet blade angle is studied in [E.C. Bacharoudis, et al, (2008)]. Theoretical, numerical and experimental investigation into prediction methods of pump as turbine performance was carried out in [Sun-Sheng Yang, et al, (2012)]. Effect of blade number and Modeling of turbulence on the pump characteristics is carried out by numerical simulation [B. Jafarzadeh, et al, (2011)]. Numerical analyses of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points were discussed in [Raúl Barrio, et al, (2010)]. A survey of literature and industrial experience provided a shortlist of key parameters in the design process and that are likely to have an effect on the head, discharge and power variation in the pump. The parametric study utilizes DOE to reduce the number of analyses required at each flow rates being investigated. The DOE [Barker Thomas B. (1985) , Kuehl Robert O (2000), Montogomery Douglas C.(2007)] and CFD provides a framework for the post-processing of the results and allows the performance variation in conjunction with the adjustment of the design variables. The objective of this work is to assist the development of pump designs, which will achieve best efficiency point without significant loss in the performance. This present paper utilizes the analysis to provide a wider parametric study that investigates the effect of various geometry features on the centrifugal pump.

PROBLEM FORMULATION

Pump Geometry

The focus of the investigation is a low specific speed domestic centrifugal water pump. The centrifugal pump simulated is of a single entry, volute type, shown in “Figure 2”, with a specific speed of 5.87. The impeller has a maximum diameter of 162 mm, with 5 backwards curved blades and splitter blades. The impeller blade has average inlet and outlet angles of 19^o and 43^o, respectively. The pump operates at a speed of 2800 rpm, with a duty flow condition of 3000lph. The geometrical factors considered for the parametric study and the design parameters [1-8] of the impeller are in "Figure 2 " and “Table 1”.

CFD Analysis

CFD tools (Computational Fluid Dynamics) are used to predict the pump performance at various conditions [1-12]. The flow characteristics such as pressure, volumetric flow, velocity, temperature at each point can be obtained from CFD tools. The overall flow pattern and velocity distribution in the whole volume of flow can be obtained in the form of graphical representation for best understanding [1-7].

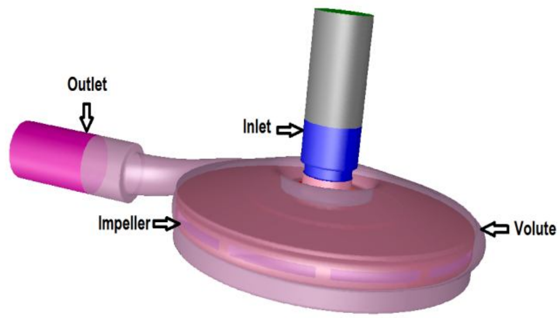

Figure. 2. Model of the pump

Table 1. Primary design parameters of impeller

S.No	Parameters	Model
1	Outer Diameter of impeller	162 mm
2	Blade angle at outlet	43^{o}
3	Blade angle at inlet	19^{o}
4	Blade width at exit	4.5 mm
5	Eye diameter of impeller	27 mm
6	Number of vanes	5
7	Number of splitter vanes	5

CFD procedure requires many assumptions to be made such as number of elements, element types, turbulence model etc., to solve a problem. As no assumption is universal in nature, it is impossible to make an assumption to a problem. Assumptions are selected by trial and error method where the assumptions are varied to match the CFD results with experimental results. This is known as Best practice of CFD. The assumptions which match the CFD results with experimental results are taken as standard for further analyses of optimization. CFD uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow (Navier-Stokes and allied equations), for predefined geometries and boundary conditions. The result is a wealth of predictions for flow velocity, temperature, density, and chemical concentrations for any region where flow occurs.

Figure.3. Impeller Mesh

Grid Independence Test

Grid convergence is the term used to describe the improvement of results by using successively smaller cell sizes for the calculations. A calculation should approach the correct answer as the mesh becomes finer, hence the term grid convergence. The normal CFD technique is to start with a coarse mesh and gradually refine it until the changes observed in the results are smaller than a pre-defined acceptable error. Here we require 4 lakh elements for surface meshing and 13 lakh elements for volumetric meshing for best results in "Figure 3 ".

Steps and Tools Involved In CFD

- Preprocessing
  - Domain Extraction (ANSA)
  - Model cleanup (ANSA)
  - Surface Meshing (ANSA)
  - Volume Meshing (TGRID)
- Solving (FLUENT)
  - Navier Stokes equation is solved in the centroid of volume mesh. Boundary conditions are applied in this stage
- Post processing (FLUENT)
  - Extraction of result
  - Graphical representation

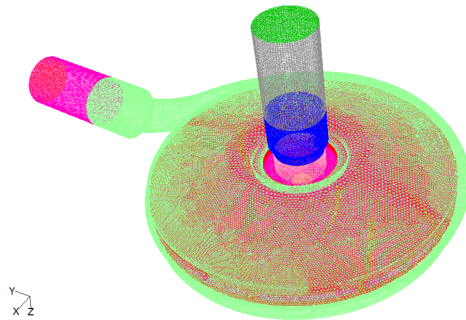

Figure.4. Surface mesh

In the preprocessing stage the fluid flow path is extracted from the 3D model of the pump. Model cleanup process involves removing unwanted surfaces, correcting overlapping surfaces and removing clearance between two surfaces. The outer surface of the fluid flow path is discretized to form the surface mesh. Surface mesh is the base for volume mesh. 4 lakh triangle elements are used in surface mesh with 0.6 skewness. Volume mesh is created from the surface mesh in "Figure 4" with 13 lakh tetrahedron elements with 0.8 skewness. The total volume of fluid flow path is discretized in this stage. TGRID is used for discretization. A trial analysis has been carried out on the meshed volume to predict the performance of the pump under the duty point condition of Volume flow rate 3000 lph and rotational speed 2800 rpm as in "Figure 5".

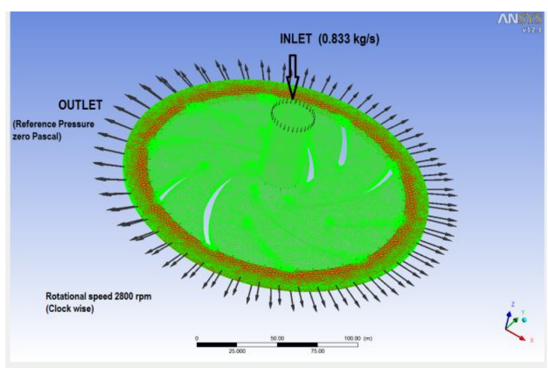

Figure.5. Boundary Conditions

Assumptions In Solving Process

Fluent 6.3 is used for solving the problem. K–ω turbulence model is chosen for analysis. First order discretization scheme is chosen for solving. As no turbulence model is universal in nature, it is required to find out the best combination of assumptions. This is done by comparing the CFD results obtained from different assumptions with the experimental results. The assumptions which match the results with the experimental performance curve are taken as standard. The result and performance curve are given in "Table 2".

RESULTS AND DISCUSSIONS

Post processing involves extraction of result at the point of interest. Graphical representation of various characteristics in the whole flow volume gives the best understanding of the solution. Navier Stokes equation is solved at the centroid of each element during the solving process. From that solution, required results can be extracted at the point of interest or at any cross sectional area.

Table 2. Experimental and CFD Results

Total Head (m)	Experimental Discharge (LPS)	CFD Discharge (LPS)
36.25	0	0.070
33.25	0.76	0.838
30.25	1.15	1.172
27.25	1.37	1.381
24.00	1.56	1.578
21.50	1.69	1.710
18.25	1.84	1.867
15.00	1.93	1.970
12.00	2.00	2.050
9.50	2.06	2.101
7.50	2.09	2.130

The results obtained for the condition 33.25 m head and impeller speed 2800 rpm are shown in “Table 3”.

Table 3. CFD Results at duty point

Characteristics	Result
Total Pressure developed	4.498 × 105 Pa
Volume flow rate	0.833 × 10-4 m3/s
Opposing torque generated by fluid	1.95 Nm
Hydraulic Efficiency	85%
Head (Suction + Delivery)	33.25 m

OPTIMIZATION USING RSM

Identification Of Predominant Factors And Finding Upper And Lower Limits Of Chosen Factors

To perform numerical simulations, which were carried out to analyze the effects of the key design parameters, including the eye diameter of the impeller, the outlet blade angle and the blade width at exit were identified. A detailed analysis has been carried out to fix the lower and upper limits of the factors. Based on the analysis, the upper and lower limits of the factors are identified as in Table 4. Experimental design (DoE) is done with the help of Response Surface Method (RSM). Experiments are designed with three factors at five levels i.e. –α, -1, 0, +1, +α. Experimental design is carried out to minimize the number of trials [15-18].

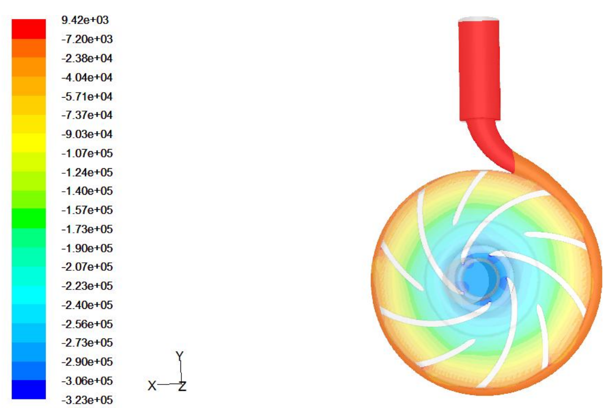

Figure.6. Static pressure contour

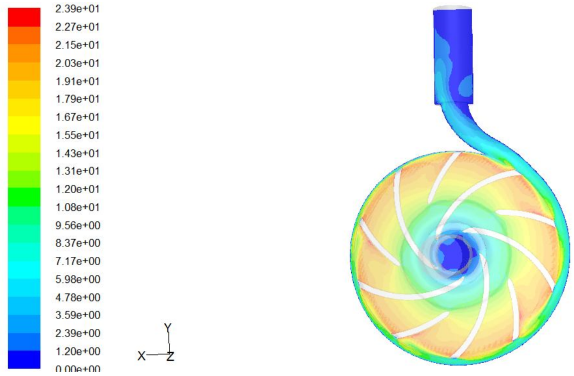

Figure.7. Velocity contour

Table 4. Design parameters and levels

Design Parameters		**Levels**				
		- 2	**-1**	**0**	**1**	**2**
A	Eye diameter of the impeller (mm)	25	26	27	28	29
B	Vane exit angle (deg)	32	37.5	43	48.5	54
C	Blade width at exit (mm)	3.5	4	4.5	5	5.5
Noise Factors		Range				
Speed (rpm)		2784-2901				
Current (amps)		4.98-5.62				

CONCLUSIONS

In this study, a steady state liquid flow in a centrifugal pump was investigated numerically and experimentally. The continuity and Navier-Stokes equations were used accounting for the k-e turbulence model and the standard wall functions. The ANSYS-FLUENT code was applied to solve numerically these equations and to perform numerical simulations, which were carried out to analyze the effects of the key design parameters, including the eye diameter, the outlet blade angle and the blade width at exit had on the pump performance. The pump overall efficiency is also influenced by the selected key design parameter. A relatively good agreement was observed comparing the developed numerical approach with the experimental results. These results are given as responses for optimization method to find out the best combination of design parameters which give the optimum performance of the pump can be found in Part II.

ACKNOWLEDGMENTS

The authors are grateful to Vipin Kumar H. Product Development, Electromechanical Division, V-Guard Industries Ltd., Cochin, India.

REFERENCES

1. M.H. Shojaeefard et al, (2012), “Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid,” An International journal: Computers & Fluids60, pp.61–70.
2. Jie Jin, Ying Fan, et al, (2012), “Design and Analysis on Hydraulic Model of The Ultra -low Specific-speed Centrifugal Pump,” International Conference on Advances in Computational Modeling and Simulation, .110 – 114.
3. R. Barrio, et al, (2012), “Estimation of radial load in centrifugal pumps using computational fluid dynamics,” European Journal of Mechanics B/Fluids 30, .316–324.
4. E.C. Bacharoudis, et al, (2008), “Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle,” The Open Mechanical Engineering Journal, .75-83.
5. Sun-Sheng Yang, et al, (2012), “Theoretical, numerical and experimental prediction of pump as turbine performance,” Renewable Energy: An International Journal, 48.
6. B. Jafarzadeh, et al, (2011), “The flow simulation of a low-specific-speed high-speed centrifugal pump,” Applied Mathematical Modelling 35, .242–249.
7. Raúl Barrio, et al, (2010), “Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points,” An International journal: Computers & Fluids 39, .859–870.
8. John S and Anagnostopoulos, (2009) “A fast numerical method for flow analysis and blade design in centrifugal pump impellers,” An International journal: Computers & Fluids 38, .284–289.
9. J. Fan, et al, (2011), “Computational fluid dynamic analysis and design optimization of jet pumps,” An International journal: Computers & Fluids 46, .212–217.
10. Wang Ji-Feng, et al, (2012), “A novel design of composite water turbine using CFD,” Journal of Hydrodynamics, .11-16.

11. H Chen, et al, (2010), "Impellers of low specific speed centrifugal pump based on the draughting technology," 25th IAHR Symposium on Hydraulic Machinery and Systems.

12. R. Spence and J. Amaral-Teixeira, (2009) "A CFD parametric study of geometrical variations on the pressure pulsations and performance characteristics of a centrifugal pump," An International journal: Computers & Fluids, 38, .1243–1257.

13. R. Spence and J. Amaral-Teixeira, (2008) "Investigation into pressure pulsations in a centrifugal pump using numerical methods supported by industrial tests," An International journal: Computers & Fluids 37, .690–704.

14. Wei Hana, et al, (2012), "The Numerical Analysis of Radial Thrust and Axial Thrust in the Screw Centrifugal Pump," International Conference on Advances in Computational Modeling and Simulation, 31, .176 – 18.

15. Punit Singh and Franz Nestmann, (2010) "An optimization routine on a prediction and selection model for the turbine operation of centrifugal pumps," Experimental Thermal and Fluid Science 34, .152–164, 2010.

16. Barker Thomas B. (1985) , "Quality by experimental design". New York: Marcel Dekker Inc..

17. Kuehl Robert O (2000)." Design of experiments". USA: Duxbury.

18. Montogomery Douglas C.(2007) "Design and analysis of experiments". New Delhi: Wiley India (P) Ltd.