IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

Performance Investigation of a Centrifugal Pump by Dimensional Modification Using CFD and Experiments

[1]Divyesh Chovatiya, [2]Ashish Vasiyar

[1]PG Student, [2]Assistant Professor,
[1]Master of Mechanical Engineering Department,
[1]Noble Group of Institute, Junagadh, India

Abstract: *Centrifugal pumps are probably among the most often used machinery in industrial facilities as well as in common life. Centrifugal pump is widely used as a universal machine, so improving performances of the centrifugal pump is important for energy saving. With the development of computational fluid dynamics (CFD), numerical simulation has become the main method for performance prediction and structure design of the pump. The impeller is the most important part in a centrifugal pump since it is the place where the mechanical energy is converted into hydraulic energy. Hence the parameters related to the impeller are directly affecting the performance of the pump. Parameters are the blade exit angle, Blade inlet angle, blade diameter etc. if it is not correctly designed, it can negatively affect the head and the hydraulic efficiency of the pump. The previous research suggests that a larger blade angle is suitable. Also, exit blades angle variation contributes more in the performance. Compared with the values of theoretical and CFD simulation, the solution of the final design point exhibits a good consistency.*

***Index Terms* – Centrifugal Pump, Computational fluid dynamics**

I. INTRODUCTION: Pumps are hydraulic machines that transform mechanical energy into hydraulic energy. Hydraulic energy is pressure energy in a different shape. The induced vortex flow theory states that an external torque rotates a given mass of liquid increasing the pressure tip and this is how the centrifugal pump works. The greater the radius of the outlet impeller the higher the pressure tip and liquid will discharge from the outlet with a high pressure head. The liquid may be raised to a high degree due to the high pressure head. Centrifugal pumps are widely used in the automotive and other sectors for a variety of applications. The significant cost and time involved with the trial and error phase of designing and testing physical designs, pump manufacturers' profit margins are limited. The numerical simulation can provide very precise knowledge on fluid behavior in a machine, and it can help engineers achieve a quantitative performance evaluation of a particular design. Improving hydraulic efficiency necessitates an inverse design process that involves evaluating a vast range of alternative designs.

Function of CFD Analysis

Computational fluid dynamics to solve fundamental nonlinear differential equations that explain fluid flow for predefined geometry and boundary condition. The outcomes of predictions for temperature, density, temperature and chemical concentration.The first step in computational fluid dynamic analysis is to create a mathematical model of the physical problem. Fluid properties were empirically modeled. Provide the issue with sufficient initial and boundary conditions.

Computational Fluid Dynamic

The geometry boundary state flow analysis in turbo machinery is complicated because the geometry is three dimensional and the flow is complicated. CFD has aided in the creation of a coherent approach to turbo machinery research and design.The actual testing of turbo machinery with precise measurement in a rotating passage is costly and in many cases impossible computational fluid dynamic simulation provided detailed flow field details.

II. LITERATURE REVIEW:

The complex internal flows through the impeller has sparked lot of interest resulting in a lot of testing. An impeller rotation speed blade number and flow rate are critical design parameters that have characteristics. The inquiry focuses primarily on the pump efficiency characteristics. The study was performed for five different flow rates at different rotational speeds with different numbers of blades while maintaining the same impeller eye diameter, blade width and blade thickness.

III. DESIGN OF CENTRIFUGAL PUMP:

Modeling and Mesh Generation

Unstructured meshes are used in to minimize the amount of exhausted creating meshes by simplifying the geometry modeling and mesh generation phase by the modeling more complex geometry by traditional multi block structured meshes and allowing to overcome flow field features. Analysis workbench was used to create the geometry and mesh of a six-bladed pump impeller domain. Unstructured mesh with cells used for the impeller as shown in figure.

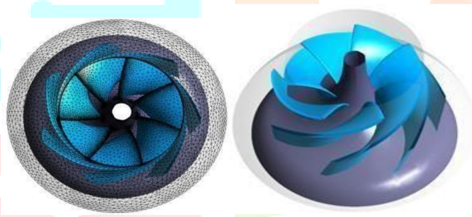


Figure: Six bladed pump impeller

Boundary conditions

The rotate speed range of 1000 to 4000 (rpm) the impeller domain of pump is considered a revolving frame of reference. The turbulence model is used and there is no slippage. The impeller blade hub and shroud have been subjected to boundary conditions. The boundary conditions are given as inlet pressure and outlet mass flow rates based on flow rates.

The centrifugal pump specification in the present study is given by Falcon Pump Pvt. Ltd. Rajkot shown in Table:

Blade thickness	2 mm
Blade width	8.24 mm
Impeller eye diameter	171 mm
Impeller outlet diameter	360 mm
Speed of the impeller	1000 to 4000 rpm
Number of blades	6, 7, 8, 9, 10

Performance Result:

The centrifugal pump impeller of different flow rates. The following table output results for five different flow rates and blades at various rotational speeds in table:

Theoretical Value :

Theoretical formula for efficiency of the pump

η=WQH/P

Where,

η= Efficiency of the pump

W= Specific weight = 9810N/m^3

Q= Flow rate (m^3/sec)

H= Heat generated

P= Shaft power

1. Theoretical efficiency for 7 blades for 22 kg/sec at 1000 rpm rotational speed

η1=(W1)(Q1)(H1)/P1

W1=9810N/m^3

Q1=22 kg/sec = 0.154 m^3/sec

H1=13.496 m

P1=21802.35 W

η1=(9810)(0.154)(13.496)/21802.35 =(20388.94/21802.35)*100=93.514%

2. Theoretical efficiency for 7 blades for 24 kg/sec at 1500 rpm rotational speed

η2=(W2)(Q2)(H2)/P2

W2=9810N/m^3

Q2=24 kg/sec = 0.168 m^3/sec

H2=36.1629 m

P2=63012.15 W

η_2=(9810)(0.168)(13.496)/63012.15 =(59599.35/63012.15)*100=93.601%

3. Theoretical efficiency for 7 blades for 32 kg/sec at 2000 rpm rotational speed

η_3=(W3)(Q3)(H3)/P3

W3=9810N/m^3

Q3=32 kg/sec = 0.224 m^3/sec

H3=64.23 m

P3=148830.64 W

η_3=(9810)(0.224)(64.23)/148830.64 =(141141.57/148830.64)*100=94.8336%

4. Theoretical efficiency for 7 blades for 34 kg/sec at 2500 rpm rotational speed

η_4=(W4)(Q4)(H4)/P4

W4=9810N/m^3

Q4=34 kg/sec = 0.238 m^3/sec

H4=107.343 m

P4=264409.0127 W

η_4=(9810)(0.238)(107.343)/264409.0127=(250622.2895/264409.0127)*100=94.7854%

5. Theoretical efficiency for 7 blades for 34 kg/sec at 3000 rpm rotational speed

η_5=(W5)(Q5)(H5)/P5

W5=9810N/m^3

Q5=34 kg/sec = 0.238 m^3/sec

H5=165.103 m

P5=408796.4799 W

η_5=(9810)(0.238)(165.103)/408796.4799=(385479.1823/408796.4799)*100=94.529614%

6. Theoretical efficiency for 7 blades for 34 kg/sec at 3500 rpm rotational speed

η_6=(W6)(Q6)(H6)/P6

W6=9810N/m^3

Q6=34 kg/sec = 0.238 m^3/sec

H6=236.966 m

P6=591210.869 W

η6=(9810)(0.238)(236.966)/591210.869=(553263.4775/591210.869)*100=93.8514%

7. Theoretical efficiency for 10 blades for 34 kg/sec at 4000 rpm rotational speed

η7=(W7)(Q7)(H7)/P7

W7=9810N/m^3

Q7=34 kg/sec = 0.238 m^3/sec

H7=294.64 m

P7=1090251.452 W

η7=(9810)(0.238)(294.64)/1090251.452=(982742.256/1090254.452)*100=90.139%

Theoretical efficiency is shown in table:

Rotation Speed(rpm)	**Number of Blades**	**Flow rate (kg/sec)**	**Theoretical Efficiency (%)**
1000	7	22	93.514%
1500	7	24	93.601%
2000	**7**	**32**	**94.833%**
2500	7	34	94.785%
3000	7	34	94.529%
3500	7	34	93.581%
4000	10	34	90.139%

Centrifugal Pump Impeller simulation by CFD

The efficiency of the backward curved blade centrifugal pump must be improved modeling and analysis. The parameterization of the impeller number of geometric variables is introduced. The results of the computations for the steady flow field in a specific impeller are analyzed using a three dimensional graph. The impeller is modeled in pro engineering software a fluid flow modeling programmer is used to perform computational fluid dynamic analysis.

Selection of pump for performance enhancement

The efficiency of pumps drops dramatically. The 3-D flow in a centrifugal volute has been numerically simulated in this study. Centrifugal pump design and performance analysis were chosen. In fluid work, the most useful rotor dynamic machine is commonly used in agriculture, industry, large plants. Pumps are used in most experimental investigations because they are limited in certain ways. To minimize the number of experiments a virtual study using package can be performed on various pump models and pump efficiency can be predicted.

Geometry Parameterization

The radial flow impeller studied in this figure 4.1 impeller built in the lab and can be represented using a small number of parameters the majority of which are shown in figure.The nominal head and volume flow rate of the impellers are determined by the rotation speed and main impeller dimension namely the exit diameter and width and exit blade angle as well as the blade inlet and exit angles.To avoid any backward effect set the exit boundary. The impeller blades are designed as circular arcs with constant width and rounded edges allowing for pump modes.The rest parameter which is a free design variable can be change to increase the impeller output and hydraulic efficiency at this specific nominal operating stage.

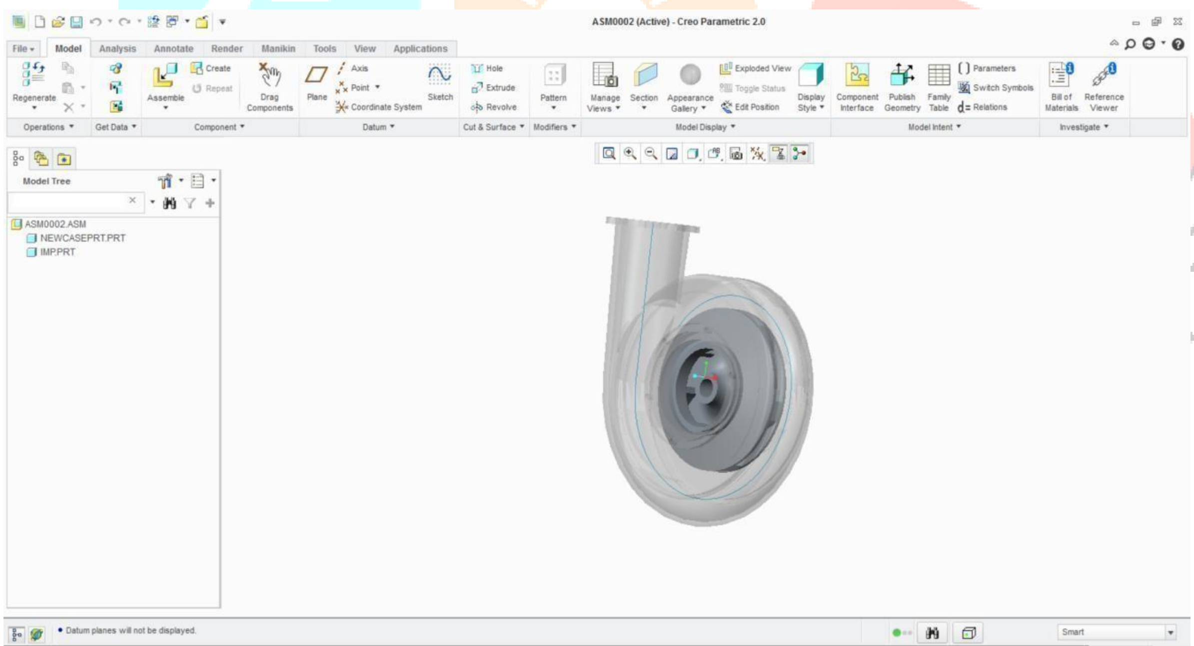


Figure 4.1 Impeller with case

Design specifications

Conventional impeller specification:

Inlet Diameter (d1) = 74 mm Outer Diameter (d2) = 200 mm Specific Speed (N) = 980 rpm Number of Vanes (z) = 6 mm

Breath of impeller (B) = 25 to 10 mm [converging from inlet to outlet]

Inlet Blade Angle (ß1) = 20 degree

Exit Blade Angle (ß2) = 24 degree

Redesigned impeller specification:

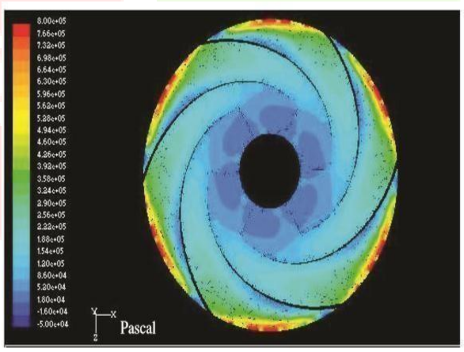
Inlet Diameter (d1) = 75 mm Outer Diameter (d2) = 230 mm Specific Speed (N) = 1500 rpm

Number of Vanes (z) = 6 mm

Breath of impeller (B) = 20.5 to 8 mm [converging from inlet to outlet] Inlet Blade Angle (ß1) = 16 degrees

Exit Blade Angle (ß2) = 23.5 degrees

Flow Analysis:



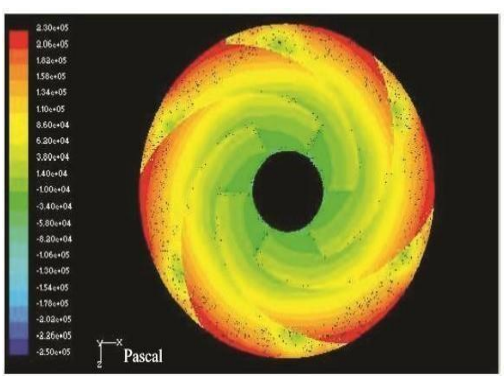


Figure 4.4 Analysis of conventionally design model

IV. RESULT AND DISCUSSION:

1. The numerical findings the number of blades is increased from 6 to 10 while flow rate is increased from 22 to 34 kg/sec. The head and efficiency are not maximized at the same rotational speed. They are not really rising or declining either. The result is best head and efficiency for various rotational speeds is calculated.

2. At 2000 rpm rotational speed 32 kg/sec for 7 blades achieves maximum performance.

3. For 6 blades at 4000 rpm rotational speed the optimal head is 22 kg/sec.

4. To simulate the problem commercial three dimensional navier-stokes was used along with a standard turbulence model.

5. The head and efficiency change regulations with regard is number of blades are complicated but there is an optimal number of blades with regard to efficiency and head separately.

6. According to the findings the current model pump optimum number of blades for best performance is 7 blades and the best head is 6 blades.

7. Different variable inlet parameters such as number of blades, mass flow rate and rotational speed are used to generate performance curves.

8. The comparison of practical and theoretical values demonstrates that the simulation is capable of accurately predicting pump characteristics.

9. Performance results Static pressure contours and total pressure contours are addressed for all flow rates with different rotational speeds for different blades.

Comparison Theoretical Efficiency and Computational fluid dynamic Efficiency:

Rotation Speed (rpm)	Number of Blades	Flow rate (kg/sec)	Theoretical Efficiency (%)	CFD Efficiency (%)
1000	7	22	93.514%	93.47%
1500	7	24	93.601%	94.54%
2000	**7**	**32**	**94.833%**	**94.77%**
2500	7	34	94.785%	94.75%
3000	7	34	94.529%	94.25%
3500	7	34	93.581%	93.53%
4000	10	34	90.139%	89.89%

With 7 blades spinning at 32 kg/sec and rotational speed of 2000 rpm maximum output or optimum efficiency is achieved.

V. REFERENCES:

1. P.Arun Kumar, P.Dhachinamoorthi, K.Saravanakumar, and S.Venkatesh, “Analysis and investigation of centrifugal pump impeller using CFD,” Department of Mechanical Engineering, Karpagam University, Coimbatore, Tamilnadu, India, and Department of Production Engineering, College of Technology, Coimbatore, Tamilnadu, India.

2. Raghavendra S.Muttalli, Shweta Agrawal, Harshla Warudkar, “CFD simulation of centrifugal pump impeller using ANSYS CFX”, P.G. Student, Department of Mechanical Engineering, MVJ College of Engineering, Bangalore,Karnataka, India,Assistant Professor,Department of Mechanical Engineering,MVJ College of Engineering, Bangalore, Karnataka, India.

3. Tilahun Nigussie,Edessa Dribssa, “Design and CFD analysis of centrifugal Pump”, Department of Mechanical Engineering, Addis Ababa Institute of Technology (AAIT), Addis Ababa, Ethiopia.

4. J.Beston, G.Gopi, S.Gopi, M.Karthika, Dr.S.V.SureshBabu, "CFD analysis of centrifugal pump in sewerage system", Department of Mechanical Engineering, Adhiyamaan College of Engineering, Anna University, Hosur, India, Professor, Adhiyamaan College of Engineering, Anna University, Hosur, India.

5. Pranav Vyavahare, Lokavarapu Bhaskara Rao, Nilesh Patil, "CFD analysis of double suction centrifugal pump with double volute",International Journal of Emerging Trends in Engineering and Development.

6. Shaji George,Boby George, Jatageetha.J, "Design and analysis of a centrifugal pump impeller using CFD as per BIS Norms", International Journal of Scientific & Engineering Research.

7. Xiangdong Han, Yong Kang, Deng Li, Weiguo Zhao, "Impeller optimized design of the centrifugal pump: A Numerical and Experimental investigation", Key Laboratory of Hydraulic Machinery Transients, Ministry of Education, Wuhan University, Wuhan 430072, China.

8. Kapil Pandya, Chetankumar M.Patel, "A critical review on CFD Analysis of centrifugal pump impeller",International Journal of Advance Engineering and Research Development.

9. Mr.Jekim,j.damor,Dilip S,Patel,Pragnesh Brahmbhat, "Experimental and CFD analysis of centrifugal pump impeller", Associate professor mechanical department GEC Modasa, Gujarat, india.

10. Aashay Desai, Umair Siddiqui, Sher Afghankhan, Emaad Ansari "Experimental analysis of performance of centrifugal pump", Mechanical Engineering Department, India.

11. P.Timar, "Dimensionless characteristics of centrifugal pump", Department of Chemical and Biochemical Engineering, Faculty of Chemical and Food Technology, Slovak University of Technology, April 2005.