ISSN 2229-5518

DESIGN AND ANALYSIS OF A CENTRIFUGAL PUMP IMPELLER USING CFD AS PER BIS NORMS

SHAJI GEORGE, BOBY GEORGE , JAYAGEETHA.J

ABSTRACT

The design of the impeller was carried out in order to improve the performance of a centrifugal pump. Design parameters from vane plane development for impeller design were selected and effect of the design parameters on the performance of the pump was analyzed using ANSYS package Computational Fluid Dynamics (CFD) is used for fluid flow analysis. Computer Aided Design (CAD) and Computer Aided Engineering (CAE) are made it possible without the actual manufacture of the impeller, a 3D model can be generated using commercial 3D CAD software's like CATIA, SolidWorks, etc. and models can be simulated for flow visualization using CFD. CFD is closer to experimentation than to theoretical fluid dynamics. Numerical simulation is done on the design specification of centrifugal pump of discharge 5 liters/s capacity, 22 m head carried out using FLUENT. Modeling of impeller is done with SolidWorks software pack and Finite element analysis is done on impeller with ANSYS software for investigation of stresses. And flow analysis is performed by using computational fluid dynamics Software. The three-dimensional flow field of the whole flow passage of a mixed-flow pump is numerically simulated by using CFD software on the basis of turbulent model according to the original design of the pump.

Through analyzing the calculation results, a new pump impeller is optimally designed. The numerically simulation results show that the hydraulic performance of the newly designed impeller of the mixed-flow pump are obviously improved, and the engineering requirements as per BIS NORMS are satisfied. The casings of centrifugal-pumps are usually made of a volute shell with two ports for supplying and removing the pumped fluid. The pump is divided along its peripheral volute path into areas with high and low flow capacity. The cross area of the volute passage of pump casings at the periphery of impeller is determined from flow capacity and on the basis of technological considerations, whereas, the wall has a constant thickness. The geometry of casings differs in details with pump type, number of stages, suction and discharge positions, and other parameters

In this study, impeller optimization is carried out to satisfy the specifications of the impeller. The CFD predicted value of the head at the design flow rate is 5.0 liters/s is 22.0 meter of water, where the pump will have optimum performance. The pressure contours show a continuous pressure rise from leading edge to trailing edge of the impeller due to the dynamic head developed by the rotating pump impeller. Near leading edge of the blade low pressure and high velocities are observed due to the thickness of the blade. The effects of the design variables are analyzed via numerical analyses, and an optimization model is suggested.

Index Terms— Computational Fluid Dynamics , Finite element analysis, Pump impeller, numerically simulated , Turbulent model , Mixed-flow pump, Hydraulic performance

SHAJI GEORGE, ME Department of M E Vimal Jyothi Engineering College Chemperi 670632
BOBY GEORGE, ME Department of M E Vimal Jyothi Engineering College Chemperi 670632
JAYAGEETHA.J,Assistant Professor,Department of ECE SNS College of Technology, Coimbatore

1. INTRODUCTION

Computational Fluid Dynamics (CFD) is the present day state-of-art technique for fluid flow analysis. In recent years the rapid development in both computer software and computer hardware has made it possible for error free design process.

ISSN 2229-5518
Computer Aided Design (CAD) and Computer Aided Engineering (CAE) are made it possible without the actual manufacture of the impeller, a 3D model can be generated using commercial 3D CAD software's like Unigraphics, CATIA, Solid Works, etc. and models can be simulated for flow visualization using CFD (Computational Fluid Dynamics). CFD is closer to experimentation than to theoretical fluid dynamics. Numerical simulation of 300 lpm capacity, 20 m head centrifugal pump carried out using commercial CFD package FLUENT is presented. The steady state simulations were carried out using Reynolds averaged Navier-Stokes (RANS) equations. It was found that non-uniformities are created in different parts of the pump at off-design conditions which result in the decrease in efficiency. The operating characteristic curves predicted by the numerical simulation were compared with the results

A pump is a hydraulic machine which converts mechanical energy into hydraulic energy or pressure energy. A centrifugal pump is also known as a Roto-dynamic pump or dynamic pressure pump. It works on the principle of centrifugal force. In this type of pump the liquid is subjected to whirling motion by the rotating impeller which is made of a number of backward curved vanes. The liquid enters this impeller at its center or the eye and gets discharged into the casing enclosing the outer edge of the impeller. The rise in the pressure head at any point/outlet of the impeller is Proportional to the square of the tangential velocity of the liquid at that point Hence at the outlet of the impeller where the radius is more the rise In pressure head will be more and the liquid will be discharged at the outlet with a high pressure head. Due to this high pressure head, the liquid can be lifted to a higher level. It has been widely used in industry is the most typical type of fluid machinery that transforms machinery energy into fluid pressure and kinetic energy via impellers.

A centrifugal pump, the most common type of pumps, has been used in industrial areas, such as water, sewage, drainage, and the chemical industry. Accordingly, numerous studies have been performed for the designs of various models of centrifugal pumps. Due to the needs of the industry, optimization using mechanical concepts has recently been studied in order to make higher-efficiency pumps with higher heads. An impeller, among all of the components of the pump, has the biggest influence on performance, since fluid flow in the pump generates energy through it. Therefore, an accurate analysis is essential to optimize variables that affect the performance of the pump. A volute gathers the outflow from the pump and delivers it to the pipe. Due to dynamic pressure loss in the case of generating internal flow, decreased pump performance can occur.

Modeling of impeller is done with **Solidworks** software pack and Finite element analysis is done on impeller with **ANSYS** software for investigation of stresses. And flow analysis is performed by using computational fluid dynamics Software. The three-dimensional flow field of the whole flow passage of a mixed-flow pump was numerically simulated by using CFD software on the basis of turbulent model according to the original design of the pump.

Through analyzing the calculation results, the reason why the flow rate of this pump can not reach to the design requirements was found out. After replacing the impeller, a new pump impeller was optimally designed. The numerically simulation results show that the hydraulic performance of the newly designed impeller of the mixed-flow pump were obviously improved, and the engineering requirements of the were satisfied. The casings of centrifugal-pumps are usually made of a volute shell with two ports for supplying and removing the pumped fluid. The pump is divided along its peripheral volute path into areas with high and low flow capacity. The cross area of the volute passage of pump casings at the periphery of impeller is determined from flow capacity and on the basis of technological considerations, whereas, the wall has a constant thickness. It should be borne in mind that geometry of casings differs in details with pump type, number of stages, suction and discharge positions, and other parameters

2. EXISTING SYSTEM

2.1 INTRODUCTION

A centrifugal pump is also known as a Roto-dynamic pump or dynamic pressure pump. It works on the principle of centrifugal force. In this type of pump the liquid is subjected to whirling motion by the rotating impeller which is made of a number of backward curved vanes. The liquid enters this impeller at its center or the eye and gets discharged into the casing enclosing the outer edge of the impeller. The rise in the pressure head at any point/outlet of the impeller is Proportional to the square of the tangential velocity of the liquid at that point Hence at the outlet of the impeller where the radius is more the rise In pressure head will be more and the liquid will be discharged at the outlet with a high pressure head. Due to this high pressure head, the liquid can be lifted to a higher level. that has been widely used in industry is the most typical type of fluid machinery that transforms machinery energy into fluid pressure and kinetic energy via impellers. A centrifugal pump, the most common type of pumps, has been used in industrial areas, such as water, sewage, drainage, and the chemical industry. Accordingly, numerous studies have been performed for the designs of various models of centrifugal pumps. Due to the needs of the industry, optimization using mechanical concepts has recently been studied in order to make higher-efficiency pumps with higher heads. An impeller, among all of the components of the pump, has the biggest influence on performance, since fluid flow in the pump generates energy through it.

2.2SPECIFIC SPEED (N_s)

ISSN 2229-5518

Specific speed is a term used to describe the geometry (shape) of a pump impeller. People responsible for the selection of the proper pump can use this Specific Speed information to :

- Select the shape of the pump curve.
- Determine the efficiency of the pump.
- Anticipate motor overloading problems.
- Predict N.P.S.H. requirements.
- Select the lowest cost pump for their application.

Specific speed is defined as "the speed of an ideal pump geometrically similar to the actual pump, which when running at this speed will raise a unit of volume, in a unit of time through a unit of head".

The performance of a centrifugal pump is expressed in terms of pump speed, total head, and required flow. This information is available from the pump manufacturer's published curves. Specific speed is calculated from the following formula, using data from these curves at the pump's best efficiency point

Specific speed (N_s) = $(NxQ^{1/2})/ H^{3/4}$

N = The speed of the pump in revolutions per minute (rpm.)

Q = The flow rate in liters per minute (for either single or double suction impellers)

H = The total dynamic head in meters

Please refer to the following chart:

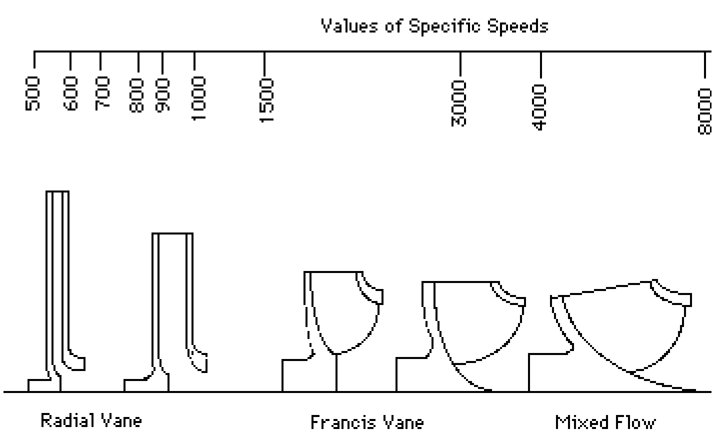


Fig 2.1

Pumps are traditionally divided into three types: radial flow, mixed flow, and axial flow. When you look at the above chart you can see there is a gradual change from the radial flow impeller, which develops pressure principally by the action of centrifugal force, to the axial flow impeller, which develops most of its head by the propelling or lifting action of the vanes on the liquid.

In the specific speed range of approximately 1000 to 6000 double suction impeller are used as frequently as the single suction impellers.

If you substitute other units for flow and head the numerical value of Ns will vary. The speed is always given in revolutions per minute (rpm.). Here is how to alter the Specific Speed number (Ns) if you use other units for capacity and head :

- Metric ....Q = M^3/hour and H = meters.
- As an example we will make a calculation of Ns in both metric and U.S. units :

- Q= 110 L/sec.
- H = 95 meters
- Speed = 1450 rpm.

2.3 EFFECT OF CHANGES IN THE IMPELLER SPEED AND DIAMETER

From dimensional analysis theory the following relationships are obtained (for high Raynolds Numbers)
Impeller speed (N in rpm)
Flow rate Q is directly proportional to N.
Differential head H is directly proportional to N^2.
Power is directly proportional to (Q X H). ie, N^3
Impeller Diameter (D)
Flow rate Q is directly proportional to D^3.
Differential head H is directly proportional to D^2.
Power is directly proportional to (QH). ie D^5.
These relationship are useful for determining the effect of changes of impeller speed or diameter for a given pump.

3. PROPOSED SYSTEM

3.1 PUMP SPECIFICATIONS

The design starts from the requirement and it is the specifications of centrifugal pump. The systematic research on the influence of the various design aspects of a centrifugal pump and in its performance at various flow rates requires numerical predictions and experiments. The specifications of centrifugal pump is given in the table 3.1

TABLE 3.1

ISSN 2229-5518

SPECIFICATIONS OF CENTRIFUGAL PUMP

Pump head H	22 m
Speed of the impeller N	2600 rpm
Flow rate, Q	300 lpm
Specific speed Ns	150 rpm

3.2.1 Design factors of impeller in centrifugal pumps

1. Let outer diameter of Impeller is D_2

Impeller velocity $U_2 = \pi D_2N/60 \quad = K_u(2gH_m)^{1/2}$

Speed ratio $K_u \quad = U_2/(2gH_m)^{1/2}$

Where, Manometric head H_m in meters and

Rotational speed of impeller N in rpm

Value of varies Speed ratio K_u from 0.95 to 1.8 depending on the specific speed.

2. Pipeline diameter: The diameter of section and delivery pipes are designed to givevelocities not exceeding1.5 to 3 m/s on section and delivery sides.

3. Discharge (Q):

The discharge or capacity of a centrifugal pump is given by

$Q = k\pi D_2B_2Vf_2$

Where k =factor which accounts the reduction in flow area due to thickness of impeller vanes, generally k=1 is considered

D_2 =Rim diameter,

B_2 =Rim width,

Vf_2=Constant velocity of flow through the impeller.

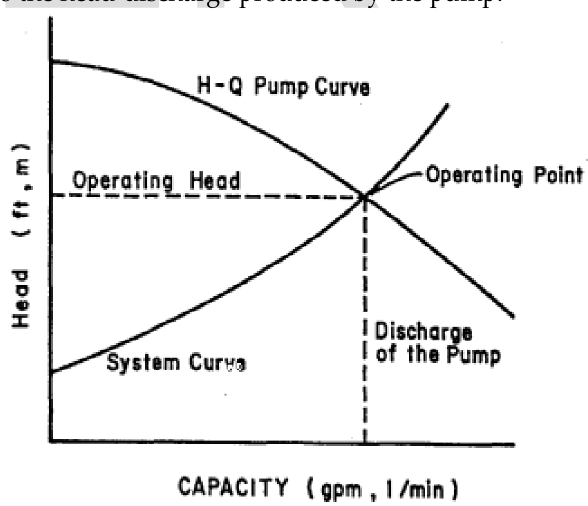
3.2.1 Design data

Item	Value
Angle of incidents, □	= 3^0 to 4^0
□	= 79^0
Outlet vane angle, □	= 27^0
No. of impeller vanes, □	□□□□
Vane pitch at inlet t_1	= 17m m
Vane thickness, S	= 4 mm
Vane blockage at inlet S_{u1}	= 4mm
Area at vane inlet, A_1	= $71.01x0^{-4}m^2$
Inlet width of impeller, b_1	= 5.2 mm
Impeller out side dia, d_2	= 150 m
Vane pitch at outlet, t_2	= 0.082 m
Vane blockage at outlet, S_{u2}	= 9.4 mm
Outlet area of impeller, A_2	= 1.05×10^{-4} m^2
Outlet width of impeller, b_2	= 1.70×10^{-3} m

3.2.2 Pump operating point

A centrifugal pump can operate at a combination System Curve of the Pump of head and discharge points given by its H-Q curve, called the operating point. Once this point is determined, brake power, efficiency, and net positive suction head required for the pump can be obtained from pump system curve

The operating point is determined by the head and discharge requirement of the irrigation system. A system curve, which describes the head and discharge requirements of the irrigation system, and a head-discharge characteristic curve of the pump which is a function of the pumping-system design. are used to determine the pump operating point At the operating point the head-discharge requirements of the system are equal to the head-discharge produced by the pump.



Determination of the operating point for a given centrifugal pump and water system.

Fig shows the determination of operating point for the typical centrifugal pump

3.4 C F D Analysis of Impeller

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements.

- Pre-processor – Geometry, Boundary conditions, Meshing
- Solver- Apply the equation solving
- Post-processor-plot generation

We briefly examine the function of each of these elements within the context of a CFD code.

3.3.1 Pre-processor

Pre processing consists of the input of a flow problem by means of an operator friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The user activity at the pre-processing stage involves,

- Definition of the geometry of the region of the interest: the computational domain.
- Grid generation: the sub-division of the domain into a number of the smaller, non-overlapping subdomains: A grid (or mesh) of cells (or control volumes or elements).
- Selection of the physical and chemical phenomena that need to be modeled.
- Definition of fluid properties.
- Specification of appropriate boundary conditions at cells, which coincide with or touch the domain boundary.

The solution to a flow problem (velocity, pressure, temperature etc) is defined at nodes inside each cell. The accuracy of a CFD solution is governed by the number of cells the better the solution accuracy. Both the accuracy of a solution and its cost in terms of necessary computer hardware and calculation time are dependent on the fineness of the grid. Optimal meshes are often non-uniform: finer in areas where large variations occur from point and coarser in regions with relatively little change. Efforts are under way to develop CFD codes with an adaptive meshing capability. Ultimately such programs will automatically refine the grid in area of rapid variations. A substantial amount of bas- development work still needs to be done before these techniques are robust enough to be incorporated into commercial CFD codes. At present it is still up to the skills of the CFD user to design a grid that is a suitable compromise between desired accuracy and solution cost.

3.3.2 Solver

There are three distinct streams of numerical solution techniques:

Finite difference, finite element and spectral methods. In outline the numerical methods that form the basis of the solver perform the following steps.

- Approximation of the unknown flow variables by means of simple functions.
- Discretization by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.
- Solution of the algebraic equations. The main differences between the three separate streams are associated with the way in which the flow variables are approximated and with the discretization processes.

3.3.3 Post processor

As in pre-processing a huge amount of development work has recently taken place in the post-processing field. Domain geometry and grid display

- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Part-le tracking
- View manipulation (translation, rotation, scaling etc)
- Color postscript output.

As in many other branches of CAE the graphics output capabilities of CFD codes have revolutionized the communication of ideas to non-specialist.

3.4Meshing in Fluent Analysis

Once the 3D Model of the Impeller is developed then instead of going for the manufacture one may go for the CFD Analysis. Such analysis will assist the manufacturer in making the manufacturing decision more quickly. The vane profile design method presented here is applied on an Impeller whose dimensions are known. This method is equally effective when used on Impellers designed using the first principles.

ISSN 2229-5518

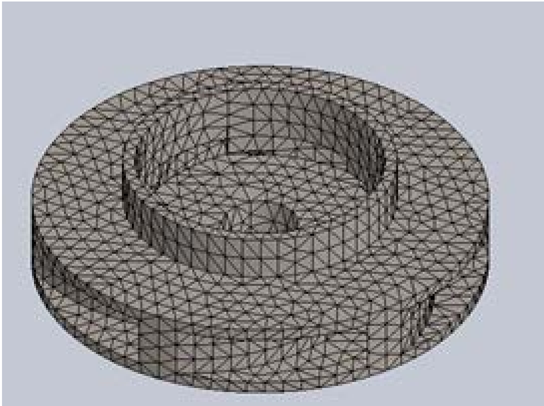


Fig 3.4.1 Unstructured mesh of the impeller

The geometry and the mesh of a six bladed pump impeller domain is generated using Ansys Workbench. An unstructured mesh with tetrahedral cells is also used for the zones of impeller. The mesh is refined in the near tongue region of the volute as

well as in the regions close to the leading and trailing edge of the blades. Around the blades, structured hexahedral cells are generated to obtain better boundary layer details. Fig. 6.3 shows the mesh near the tongue region.

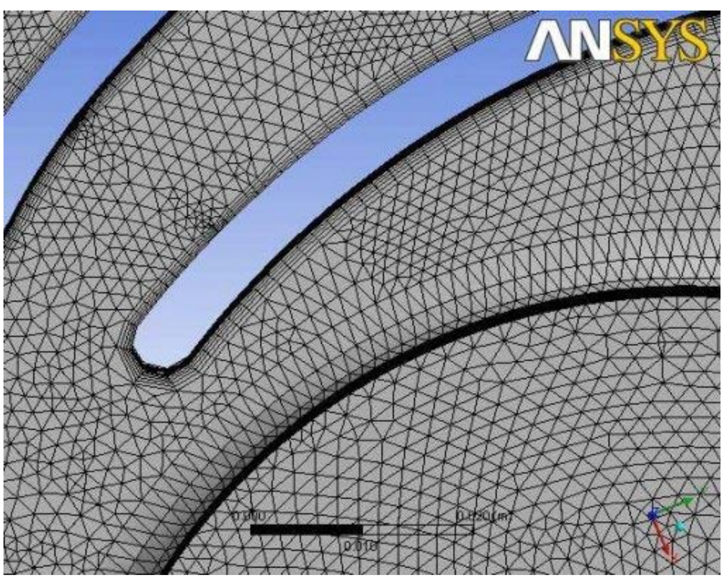


Fig 3.4.2 Mesh refinement around the blade surface

4.1 Boundary Conditions

Centrifugal pump impeller domain is considered as rotating frame of reference with a rotational speed of 2600 rpm. The working fluid through the pump is water at 25^0 C. k-ε turbulence model with turbulence intensity of 5% is considered. Inlet static pressure is 50 KPa and at the outlet and periferry ststic pressure of the impeller is 220KPa and the mass flow rate of 5.0kg/s are given as boundary conditions. Stress analysis of the impeller is shown in Figure 4.1 . The von Mises stress is used to predict yielding of materials under the above loading condition from results of simple uniaxial tensile tests. The von Mises stress satisfies the property that two stress states with equal distortion energy have equal von Mises and the stresses induced are with in the yield stress of material (stainless steel) stress.Three dimensional incompressible N-S equations are solved with Ansys-FLUENT Solver.

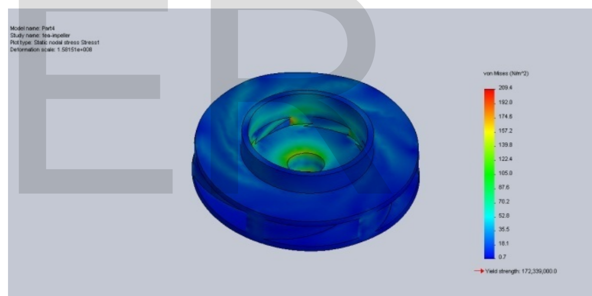
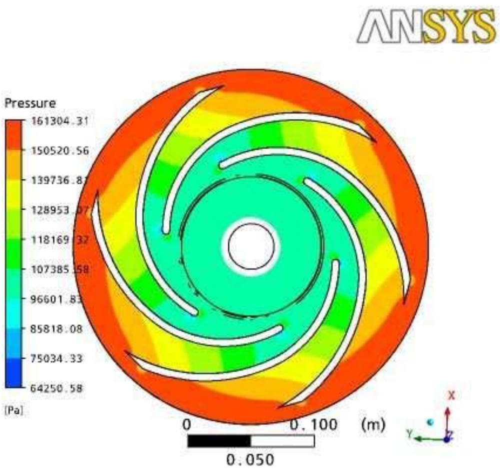


Fig .4.1 Stress analysis of the impeller

4.2Pressure Contours on A Plane

Besides the centrifugal pump performance results, the local characteristics of the internal flow field is accomplished. The Pressure contours at different span wise locations of 25, 50, 75, 90 and 100% are shown in Figures. The pressure contours show a continuous pressure rise from leading edge to trailing edge of the impeller due to the dynamic head developed by the rotating pump impeller. It is observed that total on pressure side of the blade is more than that of suction side. The difference of pressure from the pressure side to the suction side of the impeller blade is increasing from leading edge to trailing edge of the blade. The minimum value of the static pressure inside the impeller is located at the leading edge of the blades on the suction side



Mass averaged total pressure contours at leading edge show a drastic change in pressures near the blade leading edge. Variation of mass averaged total pressure contours at blade trailing edge is observed. High mass averaged total pressures on pressure side of the blade and low mass averaged total pressures on suction side of the blade are observed. Near the trailing edge of the blade total pressure loss because of wake is observed.

4.3 Rapid Proto Typing of Impeller by Using 3d Printing

4.3.1 Procedure for 3d printing

1. Create 3D model of the impeller in Solidworks Software or any CAD modeling Software.
2. It is the imported to the CAM Software using for 3D printing.
3. Align the orientation of the part to the machine table.
4. Generate the G code for fabrication
5. Load the G code for fabrication to the 3D printing machine
6. Load the material for 3D printing.
7. Start the machine for 3D printing, it is a slow process and it may take hours to print a large product.

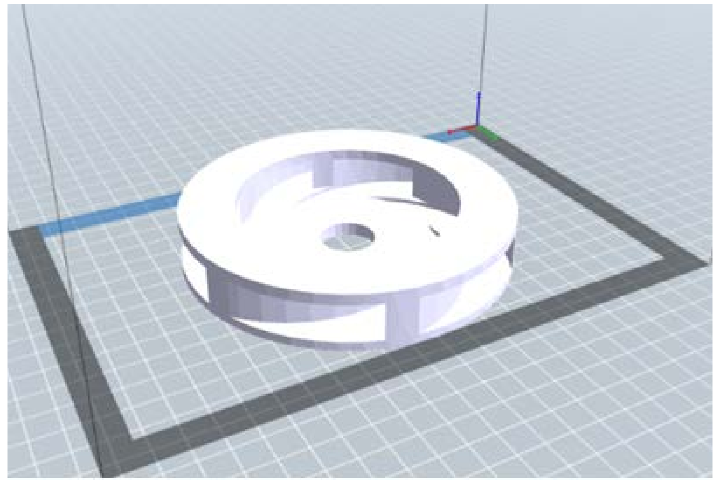


Fig 4.3 3D printing of impeller

4.4 THE PERFORMANCE OF IMPELLER

Centrifugal pump impeller with volute casing is solved at designed mass flow rate of 5.0 kg/s. The performance results are presented in Table No. The head obtained by the CFD analysis is 22.0 m and hydraulic efficiency obtained is 85.05 %.

TABLE 4.1
CENTRIFUGAL PUMP PERFORMANCE RESULTS

S. No	Parameters	Values
1	Diameter of impeller	150 mm
2	Rotational speed	2600 rpm
3	Volume of flow rate	5.0 liters/s
4	Head (in)	5.0 m
5	Head (out)	16.5 m
6	Flow Coefficient	0.0931
7	Head Coefficient	0.134
8	Shaft power	1350watts
9	Hydraulic Efficiency (at the duty point)	85.05%

CONCLUSION

A centrifugal pump impeller is modeled by using CAD Software Solidworks and CAD Model is exported to ANSYS FLU-

ISSN 2229-5518
ENT. Then mesh is generated successfully and computational flow analysis is done by using ANSYS FLUENT. The performance results are satisfactorily matching with design data and mesh quality is good. The flow patterns through the pump, performance results, circumferential area averaged pressure from hub to shroud line, blade loading plot at 25, 50, 75, and 100 % span, stream wise variation of mass averaged total pressure and static pressure, stream wise variation of area averaged absolute velocity and variation of mass averaged total pressure contours at blade leading edge and trailing edge for designed flow rate are presented.

The CFD predicted value of the head at the design flow rate is 5 liters/s approximately Head = 22.0 m. The pressure contours show a continuous pressure rise from leading edge to trailing edge of the impeller due to the dynamic head developed by the rotating pump impeller. Near leading edge of the blade low pressure and high velocities are observed due to the thickness of the blade. Near trailing edge of the blade total pressure loss is observed due to the presence of trailing edge wake.

The main points of CFD Analysis are

The increase of the designed flow rate causes a reduction in the total head of the pump.

1. Near leading edge of the blade low pressure is observed Almost similar trends were observed when the operating characteristic curves predicted by CFD.
2. The performance results show that total static head is the function of the mass flow rate with constant operating speed.
3. The model is free from cavitation at design point and it is also noted that, the formation of cavitation on the blade is increasing with the increase of mass flow rate and rotating speed

4. REFERENCES

Notations		
h_s	-	Suction head
h_d	-	Delivery head
H_s	-	Static head
H_m		Manometric head
p	-	Pressure
u	-	Tangential Velocity
A	-	Area
N	-	speed of Impeller
Q	-	Discharge
β	-	Vane Angle
ϱ	-	Density
D_2	-	Rim diameter
K_u	-	Speed ratio

1. Design of a mixed flow pump impeller and its validation using FEM analysis
By Sambhrant Srivastava, Apurb Kumar Roy and Kaushik KumarResearch Scholar, Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi, 835215 India © 2014 The Authors. Published by Elsevier Ltd

2. Design and Analysis on Hydraulic Model of The Ultra -low Specific-speed Centrifugal Pump presented in International Conference on Advances in Computational Modeling and Simulation By Jie Jina Ying Fana Wei Hana Jiaxin Hub Available online at www.sciencedirect.com Procedia Engineering 31 2012 (110 – 114)

3. Optimum design on impeller blade of mixed-flow pump based on CFD, presented in International Conference on Advances in Computational Modeling and Simulation By Jidong Lia, Yongzhong Zenga, Xiaobing Liua, Huiyan Wanga © 2011 Published by Elsevier Ltd.

4 Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid. presented by M.H. Shojaeefard, M. Tahani, M.B. Ehghaghi , M.A. Fallahian, M. Beglari Department of Mechanical Engineering, Tabriz University, Tabriz, Iran Available online 7 March 2012

5 Improving centrifugal pump efficiency by impeller trimming By Mario Šavar Hrvoje Kozmar Igor Sutlović University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture, Contents lists available at ScienceDirect Available online 6 October 2009

6 Estimation of radial load in centrifugal pumps using computational fluid dynamics by R. Barrio , J. Fernández , E. Blanco J. Parrondo a Departamento de Energía, Universidad de Oviedo, Campus de Viesques, 33203 Gijón, Asturias, Spain Available online 14 January 2011 Contents lists available at Science Direct

7. S.Rajendranand Dr.K.Purushothaman, “Analysis of a centrifugal pump impeller using ANSYS - CFX,” International Journal of Engineering Research & Technology, Vol. 1 Issue 3, 2012.

8. S R Shah, S V Jain and V J Lakhera, “CFD based flow analysis of centrifugal pump,” Proceedings of the 37th National & 4th

9. E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, “Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle,” The Open Mechanical Engineering Journal, no 2, 75-83, 2008.

10. Somashekar and Dr. H. R. Purushothama, “Numerical Simulation of Cavitation Inception on Radial Flow Pump,” IOSR Journal of Mechanical and Civil Engineering, Vol. 1, Issue 5, pp. 21-26,2012.

11 Liu Houlin, Wang Yong, Yuan Shouqi, Tan Minggao and Wang Kai, “Effects of Blade Number on Characteristics of Centrifugal Pumps,”
Chinese journal of mechanical engineering, Vol. 23, 2010.