Performance Evaluation of Centrifugal Type Boiler Feed Pump by Varying Blade Number

RAVINDRA ANANDRAO THORAT [1]

[1] M. Tech., CAMDA, MIT, Manipal

***Abstract* -** Feed pumps are an essential subsystem of boilers used in industrial process plants and called as boiler feed pump (BFP). Normally, BFP is high pressure unit that takes suction from condensate return system and can be of the centrifugal type pump. In centrifugal pump, water enters axially through the impeller eyes and exits radially. Generally, electric motor is used as prime mover to run the feed pump. To force water into boiler, the pump must generate sufficient pressure to overcome steam pressure developed by boiler. In the present study, design and analysis of boiler feed pump having a flow of 2000 m^3/hr, head of 470 m and operating at 130±10 ^oC has been taken up. The various pump parameters are obtained from design and pump model is developed using modelling software Creo Parametric. To evaluate the results at given operating conditions, CFD analysis is carried out using Ansys CFX module. Blade number has great influence on the pump performance. Therefore, CFD analyses are carried out for the pump with 5, 6 and 7 blades. Based on performance of every pump model, the best feed pump design is selected. A steady state CFD analysis is carried out using the K-ε turbulence model to solve for the Navier-Stroke's equation.

Keywords- feed pump, pump design, CFD analysis, pump performance.

I. INTRODUCTION

Pump is a machinery or device for raising, compressing or transferring fluid. It moves fluids or sometimes slurries, by mechanical action. It operates by some mechanism and consumes energy to perform mechanical work by moving the fluid. It operates via many energy sources, including manual operation, electricity, engines, or wind power. They come in many sizes, from microscopic for use in medical applications to large industrial pumps. Mechanical pumps serve in a wide range of applications such as pumping water from wells, aquarium filtering, pond filtering and aeration, in the car industry for water cooling and fuel injection etc.

Generally, pumps are selected based on required head and flow values. Pressure is required to pump the liquid through system at a certain rate. This pressure should be high enough to overcome the resistance of system, which is also called "head" [1]. The head and flow rate determine the performance of a pump. Performance curves are important to select the pump specifications.

A centrifugal pump is one of the simplest pieces of equipment in any process plant. In Figure 1 construction of centrifugal pump is shown. Liquid is forced into an impeller either by atmospheric pressure or in case of a jet pumps by artificial pressure. The blades of impeller pass kinetic energy to the liquid, thereby causing the liquid to rotate. The liquid leaves the impeller at high velocity. The impeller is surrounded by a volute. The volute converts the kinetic energy into pressure energy.

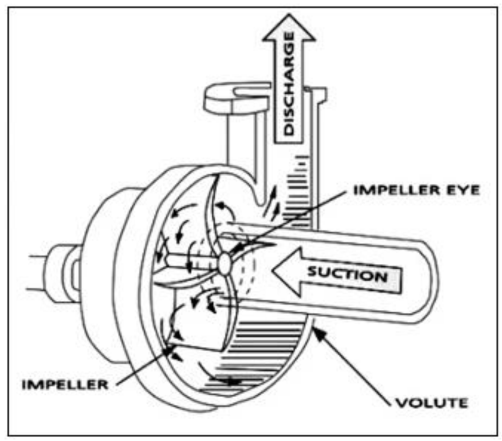

Figure. 1 Construction of Centrifugal Pump

A wide variety of centrifugal pump types have been constructed and used in many different applications in industry and other technical sectors. However, their design and performance prediction process is still a difficult task, mainly due to the great number of free geometric parameters, the effect of which cannot be directly evaluated. 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 the design and construction stage of various pump types. The rotor stator interaction can be also studied with the aid of CFD, the use of which reduces significantly the new pump development costs. The average reduction is estimated to 65% during 2005 [2]. The numerical simulation can provide quite accurate information on the fluid behavior in the machine, and thus helps the engineer to obtain a thorough performance evaluation of a particular design. The mesh generation process is a laborious task for many CFD codes, and the quality of the final mesh depends considerably on the user's experience. A Cartesian mesh approaches can also followed for some cases, where an advanced numerical technique is incorporated in order to eliminate the grid generation cost and to represent with adequate accuracy.

In recent years the sources of energy i.e. non renewable sources are going to burn out day by day. Therefore, it is required to think about use of renewable energy sources. Nuclear power generation is one renewable energy source. Boiler is used in these types of power plants [3] and therefore, boiler feed pump has the wide scope in industry.

The plan of pump system arrangement is shown in Figure 2. The main parts of system are booster pump, gear box, motor and boiler feed pump. First of all water at high pressure and high temperature comes in the booster pump. The booster pump is directly connected to motor. The water exhausted from booster pump and comes in BFP at one side. BFP runs by the motor indirectly with reduction type of gear box in between them and apply the mechanical energy on water. This mechanical energy gets converted into the hydraulic energy with the hydraulics present in centrifugal pump. The water flows out from top side of BFP and reaches into boiler.

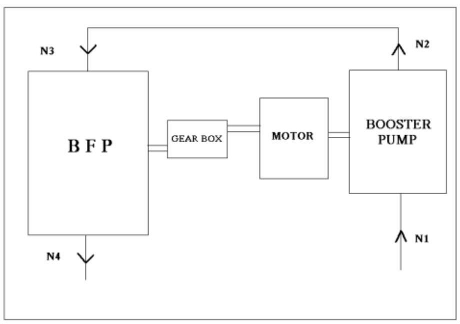

Figure 2 Plan of System

The present study aims of designing a double suction centrifugal pump having capacity of 2000 m^3/hr, head of 470 m operating at 120 ^{o}C to 140 ^{o}C and to carry out CFD analysis to evaluate its performance.

II. Design

There is no rigorous procedure to be followed in designing a pump. Lot of approaches have been developed and, although each has a slightly different method of calculation, the broad underlying principles of all are similar. The velocity limitations and proportions are also there to which it requires to adhere; but these may be exceeded in certain instances to meet competition with regard to cost or performance. The usual design is based upon a certain desired head and capacity at which the pump is operated most of time [4].

In design of centrifugal pump, the parts to be designed are: shaft, impeller, vane, casing, and selection of bearing. To design these parts different methodologies can be obtained through literature survey. From the given conditions, the specific speed is obtained [5]. According to required head, the flow rate and from specific speed, pump of double volute, double suction and single stage type is selected. The minimum shaft diameter can be obtained by using maximum shear stress theory. Impeller and vane are designed according to methodology provided by Church [6]. To design the vane empirical relations are used. API standard [7] is used to design the volute and for bearing selection. There are different methods for volute design, but "throat area from graph of ratio of throat velocity to impeller peripheral speed vs. specific speed" method is used to design a volute. The conversion of KE to PE is very important in pump and that can be achieved with the fine shape of volute. According to selection criterions stated in API standard [7], selection of bearing has been done. Specifications of feed pump are cited in table I,

Table I. Specifications of Feed Pump

Specification	Value
Head, H	470 m
Flow Rate, Q	2000 m^3/hr
Speed, N	4200 rpm
Shaft Power	2.87 MW
Temperature	120 ^{o}C to 140 ^{o}C
Pressure	6 bar
Density	1000 kg/m^3

The minimum shaft diameter is calculated on basis of strength using maximum shear stress theory. This theory predicts the yielding of ductile material. According to this theory, it is assumed that yield occurs when the shear stress exceeds the yield strength [8]. The factor of safety is assumed as 4.

The hub diameter, D_H shown in Figure 2 should be (5/16) to (1/2) times larger than shaft diameter [6]. From minimum shaft diameter, the different dimensions of stepped shaft have been finalized. The stepped shaft is designed on the basis of fitment of standard parts on shaft like; wear rings, throat bush, shaft sleeve, bearings and bearing housings, etc.

The dimensions obtained from shaft design are used in impeller design [9]. First diameter of suction flange and eye diameter of impeller are calculated. The velocity at suction flange, V_{su} assumed as 6 m/s. Suction flange diameter can be calculated with required flow and suction velocity. For a double suction pump, the leakage should not exceed 2%. Therefore, this leakage loss is considered in the flow evaluation. The eye diameter can be calculated as,

$$D_0 = \sqrt{\frac{4 \times 1.02 \times Q}{\Pi \times 2 \times V_o} + D_H^2} \tag{1}$$

For impeller inlet dimensions and angle, an inlet diameter, D_1 is taken same as D_o. Therefore, tangential velocity of inlet vane edge is given by,

$$U_1 = \omega r_1 = \frac{2 \Pi N D_1}{60 \times 2} \tag{2}$$

ISSN 2250-3153

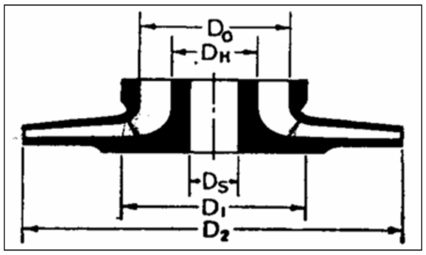

Figure. 2 Impeller Dimensions

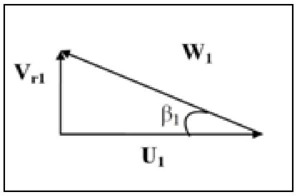

Figure. 3 Inlet Velocity Triangle

The radial velocity should be slightly higher than V_0, because a converging shape is more efficient than a divergent one. Let V_{r1} be 8 m/s.Tthe inlet angle, β_1 can be obtained from inlet velocity triangle as,

$$\beta_1 = \tan^{-1}\frac{V_{r1}}{U_1} \tag{3}$$

Usually, β_1 is increased slightly to account for contraction of the stream as it passes the inlet edges. For better performance, β_1 should be between 10^0 to 25^0 [6]. Therefore, from Figure 3, inlet radial velocity can be calculated as,

$$V_{r1} = U_1 \tan\beta_1 \tag{4}$$

The pressure head developed at the outer rim is H_2. Impeller diameter, D_2 can be calculated as,

$$D_2 = \frac{1840\sqrt{H_2}}{N} \tag{5}$$

According to tests, the head coefficientΦ is considered to account for impeller diameter. Selecting Φ = 1.01, therefore impeller diameter can be calculated as,

$$D_2 = \frac{1840\Phi\sqrt{H}}{N} \tag{6}$$

For better pump efficiency, the range for outlet vane angle is 20^0 to 25^0, therefore $\beta_2 = 21^0$. The radial velocity, V_{r2} is made same as or slightly less than radial inlet velocity V_{r1} [6], therefore V_{r2} = 11 m/s. Tangential velocity of vane at outlet can be obtained as,

$$U_2 = \omega r_2 = \frac{2\Pi N D_2}{60\times 2} \tag{7}$$

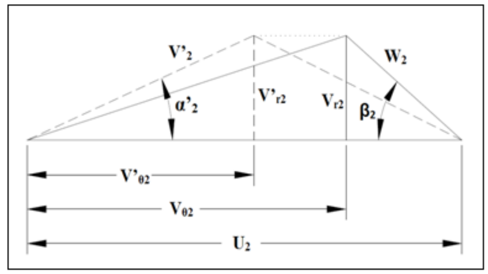

Figure. 4 Outlet Velocity Triangle

From Figure 4, the theoretical tangential outlet velocity is,

$$V_{\theta 2} = U_2 - \frac{V_{r2}}{\tan\beta_2} \tag{8}$$

Circular flow coefficient, η_θ should be considered in theoretical tangential velocity. Therefore, actual tangential velocity at outlet is given as,

$$V'_{\theta 2} = \eta_\theta \times V_{\theta 2} \tag{9}$$

From impeller design various angles, velocities and diameters are obtained. These values can be used to evaluate the pump performance. Therefore, impeller design puts the huge importance to achieve better pump performance.

Any object made of an elastic material has a natural period of vibration. When a pump rotor or shaft rotates at any speed corresponds to its natural frequency, minor unbalance will be magnified. These speeds are called as critical speeds [10]. According to API standard 610 [7], critical speed must be 20% more than system speed. Critical speed can be calculated as,

$$N_C = \frac{30}{\pi}\sqrt{\frac{g}{\delta_{st}}} \tag{10}$$

The performance of the designed pump can be studied through various efficiencies. Therefore, using velocities obtained from impeller design, various efficiencies of feed pump can be calculated.

Manometric head, $$H_m = \frac{V'_{\theta 2} U_2}{g} \tag{11}$$

Mechanical efficiency, $$\eta_m = \frac{\rho Q g H_m}{1000\times S.P.} \tag{12}$$

Overall efficiency, $$\eta_o = \eta_{man}\times\eta_m \tag{13}$$

Centrifugal feed pump model can be developed using various dimensions obtained from pump design. The values of main pump parameter are shown in Table II.

Table II. Pump Dimensions

Parameter	Value
Minimum Shaft Diameter, D_s	76 mm
Hub Diameter, D_H	114 mm
Impeller Diameter, D_2	445 mm
Inlet Vane Angle, β_1	11^o
Actual Theoretical Tangential Outlet Velocity, $V_{\theta 2}$	42 m/s
Outlet Tangential Velocity, U_2	98 m/s
Critical Speed of System, N_c	5068 rpm
Overall Efficiency, η_o	85.77%

III. CFD Analysis Of Pump

With the rapid development of the computer technology and computational fluid dynamics (CFD), numerical simulation has become an important tool to study the flow field in pumps and predict pump performance. To achieve the better pump performance CFD analysis put a huge effort. Optimized performance can be obtained by changing any of the pump parameter [11] [12].

Design of feed pump gives the dimensions of pump parts. From those evaluated dimensions, pump model is developed with the help of modelling tool. While performing CFD analysis of pump, the inlet pressure, flow rate, temperature and angular speed are provided to CFD analysis tool and analysis has been carried out. Fluid domains are required to perform the CFD analysis of pump. The fluid model can be imported in Ansys fluid flow CFX module for CFD analysis purpose [13].

The performance characteristics head and efficiency of a pump are influenced by the blade number, which is one of the most important design parameters of pumps [14]. Therefore, changing the blade numbers, CFD analyses are carried out to study the pump performances. The best and suitable blade configuration can be opted after studying the obtained pump performances. Therefore, three different models with 5, 6 and 7 number of blades are developed and those are used for analysis purpose.

The grid generation is done using Ansys ICEM CFD software which will allow the user to generate unstructured tetrahedral non-uniform mesh. A finer mesh has generated near blade, hub and shroud region where the geometry has a larger influence on the flow and where large velocity or pressure gradients were assumed to occur. To ease the discretization process, model was separated into two domains as stationary and rotating domains. Stationary domain includes inlet domain and outlet domain. Blades, hub, shrouds are included in rotating domain. The model has rotating mesh and stationary mesh region. Therefore, interface between these two regions was simulated as multiple reference frame, MRF and stage type. Based on best practices from CFX and results obtained for these two cases, it was observed that both interfaces gave similar results. MRF type interface is selected to solve in the pump simulation.The meshed model of impeller is shown in Figure 6. The head developed by the pump can be calculated by using pressure at inlet of impeller and pressure developed at outlet.

$$H = \frac{(P_1 - P_2)}{\rho g} \tag{14}$$

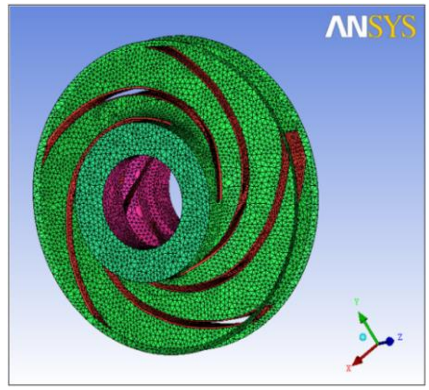

Figure 6 Meshed Impeller

After meshing, CFX-Pre is provided with input data and boundary conditions are applied to solve the problems. At inlet of pump, inlet pressure (14 bar) and at outlet of pump, the required flow (2000 m^3/hr) is provided. Density (1000 kg/m^3) as material property and operating temperature (130 ^oC) is given to fluid domain. The rotational degree of freedom, 4200 rpm is applied to impeller. Applying these boundary conditions, the problem is solved with 1000 iterations and at the end of iterations solution is converged. The analysis results for pump with 5 blades are shown in Figure 7 and 8.

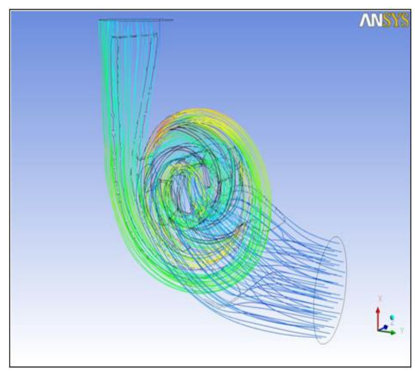

Figure 7 Velocity Streamline for 5 Blades

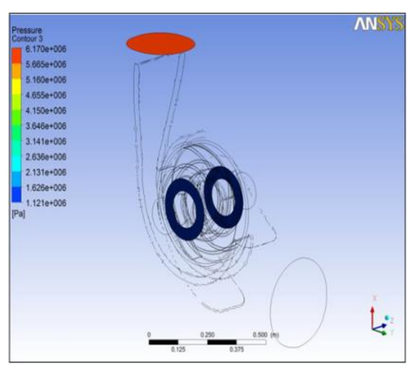

Figure 8 Pressure Contour for 5 Blades

Further, the analyses for a pump with 6 and 7 blades can be carried out. For analysis of a pump with 6 and 7 blades, the geometry is changed to 6 and 7 blades. The analysis procedure is similar as that for 5 numbers of blades. Therefore, following the same steps, the analysis results can be obtained. The input data as well as the boundary conditions are same only. Results for velocity and pressure distribution with 6 and 7 numbers of blades can be seen in Figures 9, 10, 11 and 10 respectively.

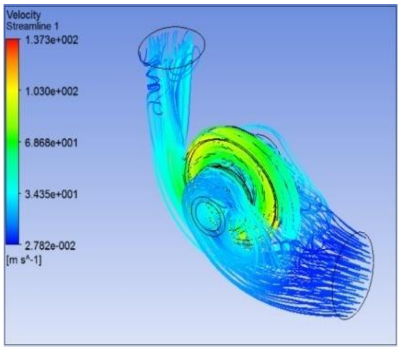

Figure 9 Velocity Streamlines for 6 Blades

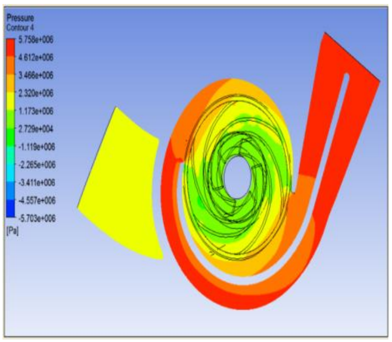

Figure 10 Pressure Contour for 6 Blades

After performing CFD analysis for feed pumps with 5, 6 and 7 blades, analysis results are obtained. These results are necessary to select the best suited pump model to fulfil the requirements. Results will show the clear picture of three cases with different impeller blades.

From CFD analyses of the pump with 5, 6 and 7 blades, various results are obtained. These analysis results are compared in Table III and Figures 13 and 14. According to results obtained from the pump analyses with 5, 6 and 7 blades, it can be observed that the streamline flow of a pump with 5 blades is smoother than that with 6 and 7 blades. In case of a pump with 6 and 7 blades, the flow streamlines are mixing at outlet. From the pressure contours for a pump with 5, 6 and 7 blades, it can be observed that the pressure at inlet to impeller is reducing. If the pressure at inlet will so much low then there may be chances of cavitation in the pump. Therefore, cavitation point of view, the pump with 5 blades is at safer side.

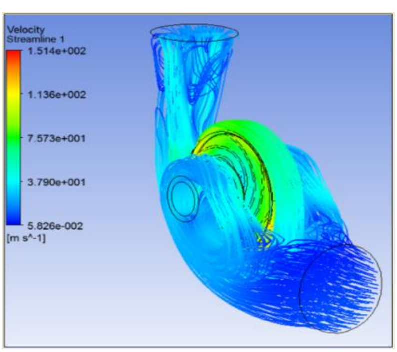

Figure 11 Velocity Streamlines for 7 Blades

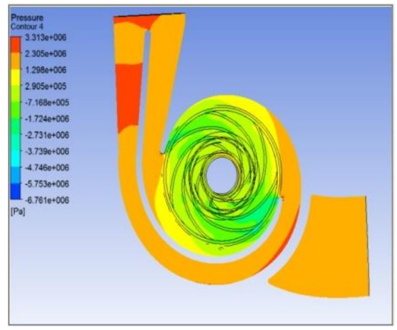

Figure 12 Pressure Contour for 7 Blades

IV. Results And Discussion

The value of required head is 470 m and proposed efficiency of the system is about 86%. For a pump with 5, 6 and 7 blades, the curves like flow versus head and flow versus efficiency can be plotted from evaluated CFD results. Head obtained for a pump with 5 blades is closer to the required head compare to other two blades. Efficiency obtained for 5 blades is higher than other two blades.

Table III Results-Flow vs. Head

No. of Blades	Head, m
5	475.17
6	476.20
7	481.40

The limitation of space between blade and flow stream gets increased with increase in blade number. The area of low pressure region at the suction of blade inlet grows continuously. With increase of the blade number, total pressure in the region of flow grows continuously. The head of centrifugal pump grows all the time with the increase of blade numbers and total pressure too, but the change in hydraulic efficiency with variation in blade number is complex.

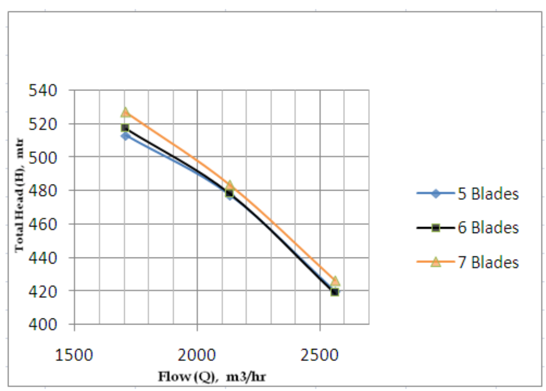

Figure 13 Flow vs. Head

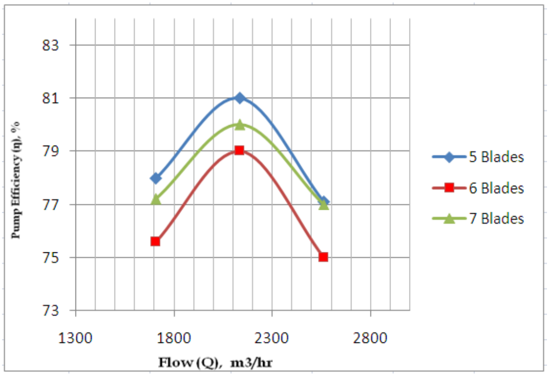

Figure 14 Flow vs. Efficiency

If the blades are too more, the crowding effect phenomenon at the impeller is serious and the velocity of flow increases, also the increases of interface between fluid stream and blade will cause the increment of hydraulic loss [14]. The critical speed is inversely depends upon the shaft deflection. Therefore, if the shaft deflection is more, lower the critical speed. The machine should not fall below the permissible limit of critical speed. Therefore, critical point of view, the system is on safer side with 5 blades. The pump model with 5 numbers of blades provides the better performance, therefore it can be selected as best performing model and the analysis results are obtained.

V. Conclusion

Some conclusions on the design and CFD analysis of centrifugal type feed pump are,

- The dimensions recommended for all parts of the pump are meeting the design requirements.
- CFD analysis shows that, feed pump with 5 blades has the best performance compared to 6 or 7 blades.

References

[1] Christian Allerstorfer, "Bachelor Thesis Centrifugal Pumps", Alma Mater Leobeinses University, Leoben.

[2] John Anagnostopoulos, "CFD Analysis and Design Effects in a Radial Pump Impeller", Wseas Transactions on Fluid Mechanics, Volume-1, Issue-7, July-2006, PP. 763-770.

[3] H Sedaghati Nasab, M Massoumian, H Karimi, "CM Experiences with Boiler Feed Pump", Journal of Worlds Pump, January-2011.

[4] Greg Case, William Marscher, "Centrifugal Pump Mechanical Design, Analysis and Testing", Proceedings of 18th International Pump User's Symposium, PP. 119-134.

[5] R P Horwitz, "The Affinity Laws and Specific Speed in Centrifugal Pumps", 14th Technical Information Bulletin, 2005, PP. 1-5.

[6] Austin H Church, "Centrifugal Pumps and Blowers", J Willey, John Willey and Son's Publication, Michigan, 1944.

[7] "Centrifugal Pumps for Petroleum, Petrochemical and Natural Gas Industries", ANSI/API Standard-610, 7th Edition, API Publishing Services, Washington, September-2010.

[8] C R Mischke, "Design of Shaft", Shaft and its Design Based on Strength, Module-8, Lesson-1, Version-2, 1989.

[9] Khin Cho Thin, Mya Mya khaing and Khin Maung Aye, "Design and Performance Analysis of Centrifugal Pump", World academy of Science, Engineering and Technology, 2008, PP. 422-429.

[10] Igor Karassik, J Messina, P cooper, C C Heald, "Pump Handbook", 3rd Edition, McGraw Hill Publication, New York, 2001.

[11] S R Shah, S V Jain, V J Lakhera, "CFD for Centrifugal Pumps: A Review of the State-of-the-Art", Chemical, Civil and Mechanical Engineering Tracks of 3rd Nirma University International Conference, Ahmedabad , india, 2012, PP. 715-720.

[12] M Mohammed Mohaideen, "Optimization of Backward Curved Aerofoil Radial Fan Impeller using Finite Element Modelling", 2012, PP. 1592-1598.

[13] S R Shah, S V Jain, V J Lakhera, "CFD Based Flow Analysis of Centrifugal Pump", 4th International Conference on Fluid Mechanics and Fluid Power, Chennai, December-2010, PP. 435-441.

[14] S Chakraborty, K M Pandey, "Numerical Studies on Effect of Blade Number Variations on Performance of Centrifugal Pumps at 4000 rpm", International Journal of Engineering and Technology, Volume-3, Number-4, August-2011, PP. 1-7.

Authors

First Author – Ravindra Anandrao Thorat, MTech, CAMDA, Manipal Institute of Technology, Manipal, ravindrathorat89@gmail.com.