CFD analysis of effect of mass flow rate and impeller speed on the performance of a centrifugal pump

[1]Shishir Kumar, [2]Ashish Muchrikar
[1]Student, [2]Assistant Professor
Corporate Institute of Science and Technology, Bhopal, India

***Abstract* - In the present study a three dimensional centrifugal pump is analyzed by CFD analysis by using ANSYS FLUENT. Three different value of mass flow rate is taken for inlet condition as 1.0345, 0.667 and 0.3846 kg/s and at each mass flow rate three different rotational speed of impeller is taken as 2400, 2600 and 2800rpm respectively. All the cases are simulated for a fixed blade number of 7. The study was aimed to analyze different performance parameters and cavitation for varying mass flow rate and change in rotation speed of impeller for a fixed blade number. The result obtained is validated with existing work and found in good agreement with them. The maximum efficiency in all cases occurs at 1.0345 kg/s mass flow rate and 2800 rpm After analyzing the water vapor volume fraction contour , it is found out that at fixed inlet and outlet pressure the probability of occurrence of cavitation is less as we decrease the mass flow rate. Also at fixed mass flow rate increasing the rpm increase the water vapor volume fraction which means chances of cavitation is more on increasing the rpm. No cavitation is seen at 0.3846 kg/s mass flow rate for all rpm while a slight water vapor volume fraction is observed at 0.667 kg/s at 2800rpm which increases at 1.0345 kg/s as we increase the rpm, representing the worst condition for the blade erosion and serve breakdown of the operation of centrifugal pump in terms of cavitation. Also pressure coefficient and head coefficient is discussed in the study for varying mass flow rate and varying impeller speed**

***keywords* - computational fluid dynamics, centrifugal pump, impeller, cavitation, efficiency, head coefficient, pressure coefficient**

I. INTRODUCTION

Centrifugal pumps are the devices which increases the pressure energy of the fluid by utilizing the mechanical energy supplied. They may be single or multi stage reliant on the number of the impeller. At contemporary there are several types of centrifugal pump presented in the arcade with single and double entry, with multiple rotor stages. Efficiency of the pump is influenced by the application. These types of pumps are utilized for many application and able to control liquid and gases at relatively high pressure and temperature. Rotor and volute are the two crucial element of the pump. The section which provides energy to the fluid generally named as impeller is called rotor and the section around which fluid displaces is named casing. Efficiency of the pump is influenced by the construction of the impeller. Shape of specific flow be influenced by the model construction of the pump. Recirculation and separation may occur at part flow conditions and due to the formation of the vapor bubbles, cavitation occur. Because of the impeller rotation, blade unsteadiness present in centrifugal pump which goes through the static volute cutwater and diffuser blade. Transient effects on the off design situation and according to time variation, it affects the change in mass flow through the pump.

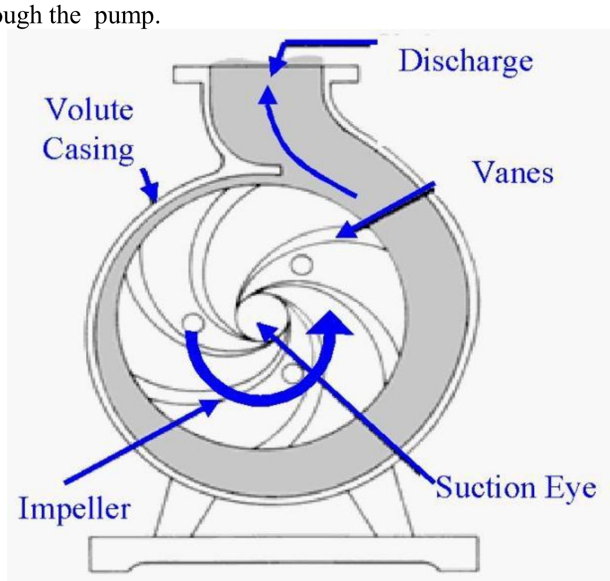


Figure 01: Centrifugal Pump

The main section of the pump which is impeller and casing is hard to design because of various geometrical considerations. To supply the energy competently to the fluid, the topographies of the impeller such a number of blades, inlet and outlet blade angle, leading and trailing edge, eye diameter must be designed prudently and specified. The volute section of the pump may be round or spiral and cross-sectional area must be round, trapezoidal or rectangular shapes. Main components of centrifugal pump are Impeller, Casing, Suction pipe and Delivery pipe.

II. LITERATURE REVIEW

Minggao et al. [1] have investigated the numerical research on recital forecast for these pumps. Commercial FLUENT software with standard k-ε algorithm was used to simulate the presentation of several centrifugal pumps design at various mass flow rate and off design flow rate for the improvement of performance and numerical calculation method. Every pump made run at different specific speed. They found an increment in the static pressure on diffusion section at the volute outlet due to the small quantity of the flow rate while at big flow rate it decreases on the similar place. Incident angle is negative for big flow rate and positive in small flow rate.

Karanth et al. [2] carried out the computational analysis on the influence of fluctuating number of diffuser slats on impeller-diffuser flow interaction in a centrifugal fan. FLUENT 6.3 with standard k-ε turbulence set-up was employed for numerical analysis. They realized that the static pressure recovery is better in odd number of diffuser than the even number. In volute casing, static pressure reduction occurs when number of diffuser increased. Amplitude of static pressure variations decreases at the outlet flange when number of diffuser vanes increase.

Bacharoudis et al. [3] investigated the parametric observation of the considered pump by changing the outlet blade angle. Simulation is performed using ANSYS FLUENT CFD Code with standard k-ε model. During the examination of the impeller performance, blade outlet angle changed. The recital curve became smoother and flatter due to enhancement in the blade angle at all span of the flow rates. Due to increase in outlet blade angle from 200 to 500, the increment in the head is more than 6 %. They obtained that at enhanced flow rate, change in the head is more due to increase in the blade outlet angle. Hydraulic efficiency also upsurges due to increase the outlet blade angle.

Ozturk et al. [4] have examined the influence of impeller-diffuser radial break ratio in centrifugal pump. The multi-aimed FLUENT with standard k-ε RNG algorithm was utilized for simulation. A five blade impeller with backward curved and nine vaned diffusers were used running at 890 rpm. Different clearence 10%, 15% and 20% was used and pressure instabilities is maximum for 20% radial gap. They obtained that along front half of suction side, a significant increment is done in pressure instabilities due to reduction in radial clearence impeller blade and diffuser vane. Pressure instabilities are maximum at the trailing edge of the impeller blade.

Houlin et al. [5] have analyzed the impact of blade count on the performance characteristics of centrifugal pump using CFD code FLUENT. During the simulation study number of impeller blade varies from 4 to7. They have analyzed the flow characteristics inside the pump with cavitation and non-cavitation conditions. As the number of blade increase, head of the pump upsurges but the efficiency and NPSHR value decreased. They observed the impeller with 5 blades has the maximum efficiency.

Aman et al. [6] have investigated virtual simulation and enactment forecast of centrifugal pumps using CFD technique. They have used commercial CFD code ANSYS FLUENT 6.4 to simulate the six backward curved blade centrifugal pump using Standard k-ε turbulence modeling scheme. Flow was visualized in the centrifugal pump using 2-D computational analysis of turbulent fluid flow, including pressure and velocity distribution. They examined that at small flow rate, blade inlet has the low pressure area at the suction side. As the flow rate upsurges, area gets closed at middle of the suction side of blade. Static pressure at small stream rate upsurges on dispersal section of the volute outlet while gets diminished at greater stream rate.

Jafarzadeh et al. [7] analyzed virtual computational discussions of a low-specific-speed high-speed centrifugal pump. They have used commercial CFD code ANSYS FLUENT with Standard k- ε, RNG and RSM three turbulence modeling scheme for computational analysis to study the most suitable turbulence model. Flow was visualized in the centrifugal pump using 3-D computational analysis of turbulent fluid flow, including pressure and velocity distribution. Blade counting varies from 5 to 7 and upshot of counting of blades was observed on the efficiency of the pump. They obtained that head coefficient was maximum for 7 number of blade impeller.

Yuan et al. [8] have studied the flow within auxiliary-impellers of centrifugal pumps. ANSYS FLUENT with S.A turbulence algorithm was used for numerical analysis. They obtained that the dynamic seals of auxiliary impeller influenced to flow through volute and main impeller. The supplementary impeller centrifugal pump has high head than the pump without auxiliary impeller but it has lower efficiency. As the flow rate upsurges, the difference of head becomes large but the difference of efficiency becomes smaller.

Yang et al. [9] have studied the flow distribution in the volute section of pump using CFD code FLUENT. The computational analysis study carried out with k- ε turbulence modeling scheme. The flow distribution was evaluated with different cross-section shape of the volute, throat area of volute and radial clearence volute tongue and impeller. When throat area upsurges, pump efficiency drops in very small quantity. The maximum efficiency of the pump observed with round volute shape and spiral volute area.

Shojaeefard et al. [10] studied the influence of varying the flow passage area of the impeller using CFD code FLUENT. The FVM method was employed for discretization of the equation. The computational analysis study was performed using k -ε and SST turbulence modeling scheme. They realized that the increasing of flow passage area of the impeller from 17 to 21 mm. The head and hydraulic efficiency upsurges due to decrement of the friction losses.

Chakraborty et al. [11] studied the effects of number of blade variations on the centrifugal pumps performance at different rotational speeds using CFD code FLUENT. The number of blade varies from 4 to 12. The computational analysis study performed at different operating speed 2900, 3300 and 3700 rpm. They realized that pump head and efficiency upsurges due to upsurge in rotational speed. Due to upsurge in the number of blade, total head developed by the centrifugal pump upsurges but the efficiency decreases. The ideal number of blade was for the specific design found 10.

Chakraborty et al. [12] analyzed that with variation of number of blade what changes on the performance of centrifugal pump. Number of blades varies from 5 to 7. Study mainly focus on the efficiency of the pump and evaluated at 3000 rpm with the help of ANSYS FLUENT 6.3 software with k-ε turbulence modeling scheme. They realized that the centrifugal pump efficiency changes according to the blade number and it has maximum value for 7 blade number. Static pressure moderately upsurges from impeller inlet to the outlet. Due to upsurge in blade number, Static pressure upsurges all the time at the outlet of the volute.

Ozturk et al. [13] have studied the influence of impeller-diffuser radial gap ratio in a centrifugal pump. ANSYS FLUENT software with standard k- ε RNG algorithm was used. Nine vaned diffuser and five backward curved blade centrifugal pump was used which run at 890 rpm. At different flow rate results were represented for three different radial clearence (10%, 20%and 30%). Finally, Results were more accurate at 20% radial gap for pressure instabilities. They obtained that due to decrement in radial clearence impeller blades and diffuser vanes, an important rises in the pressure instabilities along the front half of vane of suction side. The magnitude of these fluctuations reached to peak point, when volume flow rate is large. Blade pressure instabilities were largest at trailing edge of the impeller blade.

Hussein et al. [14] have investigated the influence of rotational speed fluctuation on the static pressure in the centrifugal pump. ANSYS/ FLUENT software with standard k-ε turbulence modeling scheme were used to analyze the influence of rotational speed on the static pressure. The study was done with five twisted blade centrifugal pump. A single blade passage gives the more fine results for the static pressure contour. They realized that at high rotational speed, the static pressure contour gives negative low static pressure at the suction side of the blade, hub and shroud. Pressure reduces at the suction side when rotational speed upsurges.

III. OBJECTIVE

In the present study a 3 dimensional pump geometry is being analyzed by CFD analysis. The specified objective of the ongoing study is as follows:

1. To perform a 3 dimensional CFD analysis of a pump with rectangular volute size and validation of the results obtained in the study with the existing literature.
2. To observe the change in hydraulic efficiency in the centrifugal pump system by varying the mass flow rate at the inlet of the centrifugal pump.
3. To observe the various non-dimensional parameters involving the centrifugal pump flow such as pressure coefficient, loss coefficient etc.
4. To observe whether the cavitation is involved in the flow or not and to visualize the volume fraction of water vapor for simulating the cavitation flow.
5. To plot the various non-dimensional curves for the pump analysis.
6. To visualize he contours of various properties involved in the flow field.

IV. METHODOLOGY

In the present work a CAD model of centrifugal pump is created using Bladegen and Design modular designing software. For creating the model dimension were considered from base paper. Computational fluid dynamic analysis have been performed for all nine models at different air mass flow rate and different RPM are as follow:

CASE	MASS FLOW RATE (kg/sec)	RPM
Case 1	1.0345	2400
Case 2	1.0345	2600
Case 3	1.0345	2800
Case 4	0.667	2400
Case 5	0.667	2600
Case 6	0.667	2800
Case 7	0.3846	2400
Case 8	0.3846	2600
Case 9	0.3846	2800

CAD geometry of centrifugal pump: Three dimensional CAD model of Centrifugal pump is created using the Bladegen and design modular software with dimension from base paper, A three dimensional views of Centrifugal pump is shown in figure :

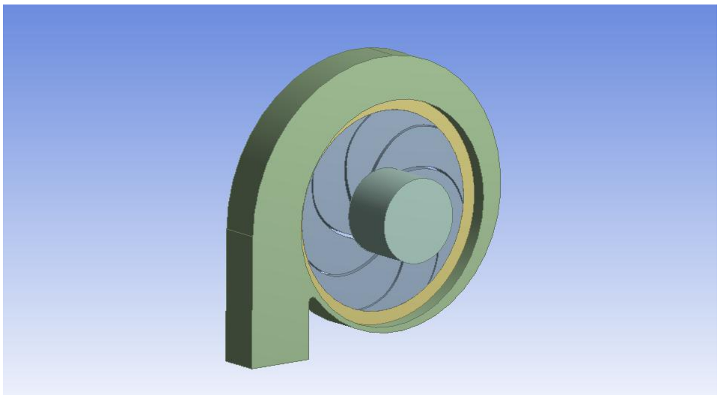


Figure 02: Three dimensional CAD geometry of model (Isometric view)

Studied pump: A small scaled CAD design with modest geometry is employed in the ongoing study (Fig. 4.4) to asses the effect of change in counting of blades. The specification of the impeller is tabulated. The gap clearance between the impeller and the volute, Dr= 7.5 mm and width b = 6 mm, is considered as vaneless diffuser. The volute section of the virtually simulated small scaled pump has a rectangular cross section with constant width 22 mm and entails of four round fragments with different radii (Fig. 4.4): RI = 66.28 mm, RII= 66.28 mm, RIII= 78.19 mm, RIV = 84.01 mm.

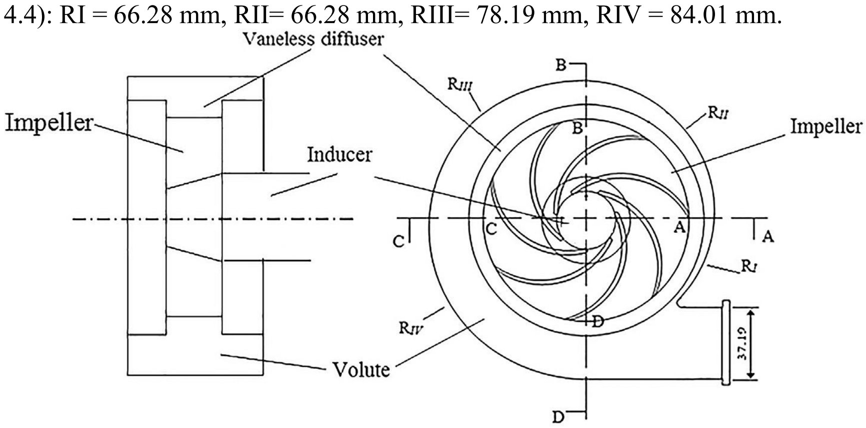


Figure 03: View of the selected pump with main dimensions.

Table 01: Impeller specifications.

Parameter	Value
Inlet diameter (D1)	45 mm
Outlet diameter (D2)	105 mm
Inlet blade width (b1)	6 mm
Outlet blade width (b2)	6 mm
Inlet blade angle (β_1)	26°
Outlet blade angle (β_2)	22°

Meshing: After completing the CAD geometry of model of Centrifugal pump is imported in ANSYS mechanical for further computational fluid dynamics analysis where next step is meshing. Discretizing the domain in small control volume is called meshing. its quality affects the results. The total number of nodes generated for this model is 314824 and total number of Elements is 335885. Types of elements used are tetrahedral which is triangular in shape with four nodes on each element.

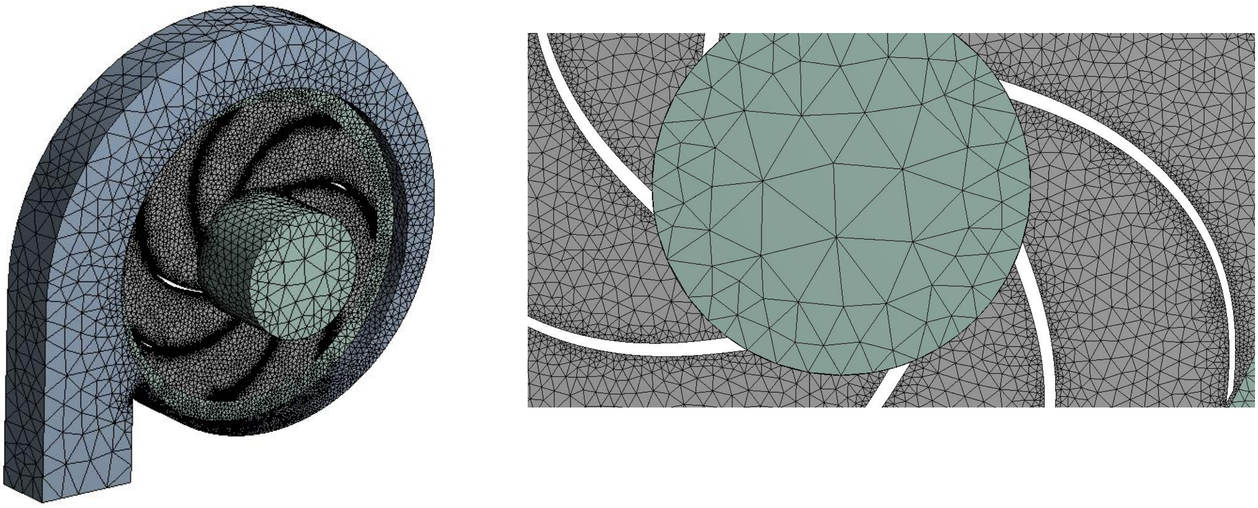


Figure 04: Meshing of CAD model isometric view and its zoomed view.

Boundary conditions are assigned to create a virtual environment of the real life working of the system. The boundary conditions for simulation of centrifugal pump are explained below:

- Define the solver settings as pressure based and enable gravity option in y direction with the value of -9.81 m/s^2.
- Working fluid is water and air with density 1.22 kg/m^3 for air and 998.2kg/m^3 for water, viscosity 1.789 e-05 kg/m-s for air and 0.001003 kg/m-s for water, Thermal conductivity 0.24 W/m^2-K.
- Set viscos model as turbulent by using RNG $k-\epsilon$ turbulence model and near wall treatment should be standard.
- Turn on multiphase model and select mixture model for simulating slug flow.
- Surface tension coefficient between air and water is 0.072n/m. .
- Set the cell zone condition for the rotating domain as per case using reference frame.
- Set the mass flow rate as per the cases for inlet of water and set inlet for air equal to zero.
- For the operating condition the operating pressure needs to be set as 0 and gauge pressure should be 111325pa.
- Under Discretization, select scheme for COUPLED, and SECOND ORDER for Momentum and Energy equation.
- The Fluent solver is used for CFD analysis.

V. RESULTS AND DISCUSSION

In the present work computational fluid dynamics analyses have been performed for Centrifugal pump which is of standard size using ANSYS fluent to investigate the effects of better performance by varying of rotation per minute of impeller and mass flow rate of inlet. For that CAD model of centrifugal pump is created using the blade gen and design modeler software with dimensions available in base paper. There are following validation and results have been discussed using contours diagram and graphical representation.

Validation : The main objective of the present work is to perform computational Fluid Dynamics analysis to maximize the pump efficiency . For the validation of this work the CAD dimensions of the centrifugal pump is taken from a research paper of Gamal R.H. Abo Elyamin "Effect of impeller blades number on the performance of a centrifugal pump" Alexandria Engineering Journal (2019) 58, 39–48, contents available at science direct. The geometrical parameters for the pump all components are considered from base paper. the working fluid used in simulation is water liquid and water vapor having properties such as density 988.2 Kg/m3 for water liquid, viscosity 0.001003 kg/m-s for water liquid, 0.01927 Kg/m3 and 8.8e-6 kg/m-s for water vapor . Types of elements used are tetrahedron which is triangular base pyramid in shape with four nodes on each element.

Computation fluid dynamics analysis for base models: Computational fluid dynamics analysis of pump for base model at all three mass flow rate 0.3846 kg/s, 0.667 kg/s and 1.0345 kg/s is done and then comparative graph is generated.

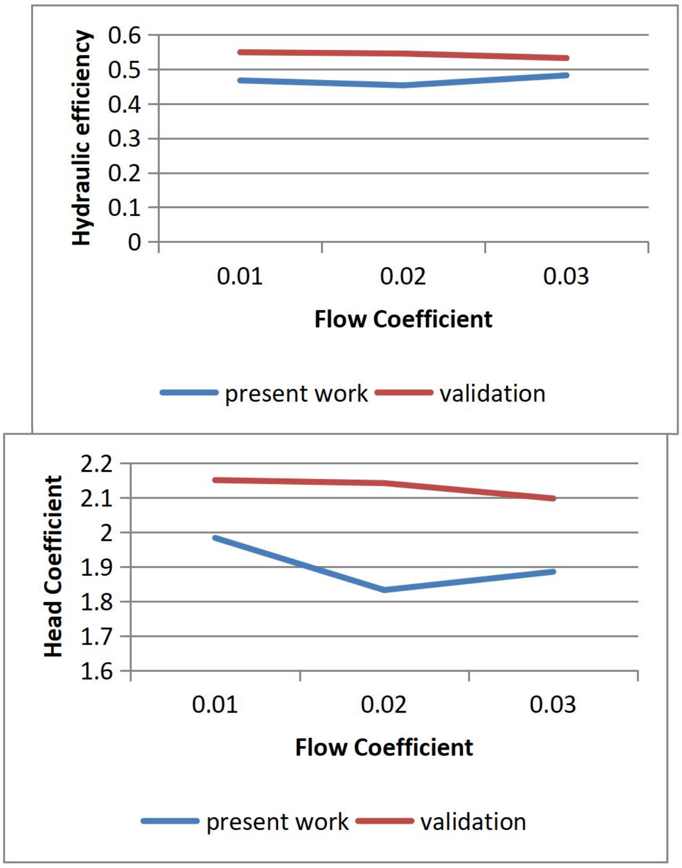


Figure 05 : Hydraulic efficiency & Head coefficient from base paper and present work

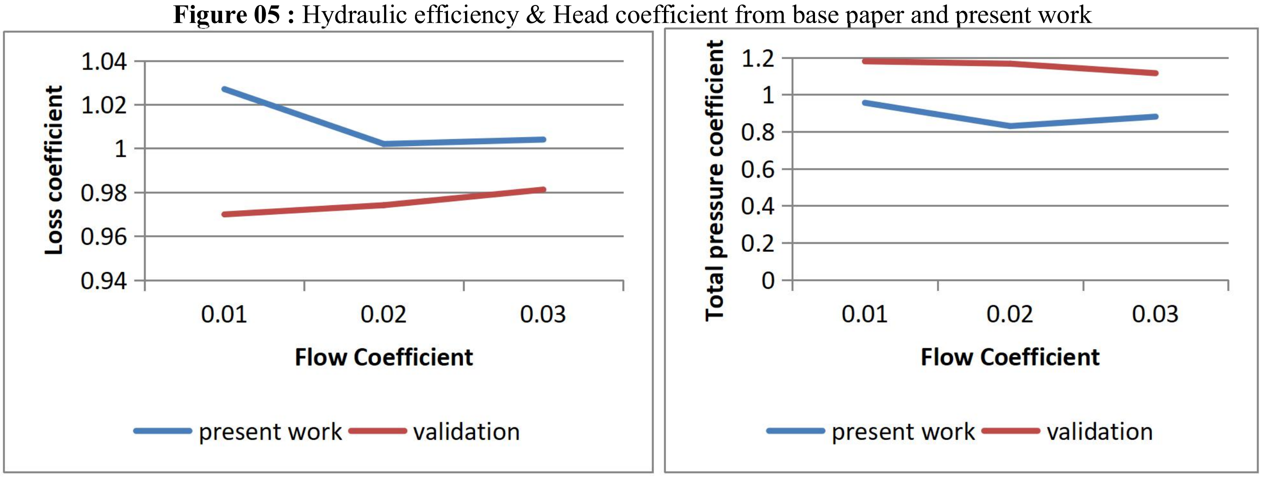


Figure 06 : Loss coefficient & Total pressure coefficient from base paper and present work

After the validation of base model some other cases of pump have been used for computational fluid dynamics analysis to investigate better efficiency and cavitation effect in pump. In the present work it has been try to improve pump efficiency by varying rotation per minute of impeller at different mass flow rate and at the mean while also capturing the influence of cavitation on the model

Results for CFD analysis of pump for all cases: After performing computational fluid dynamic analyses with absolute velocity formulation using pressure based solver. Different parameter distribution inside the pump is observed and is presented in form of contour and graph as shown below:

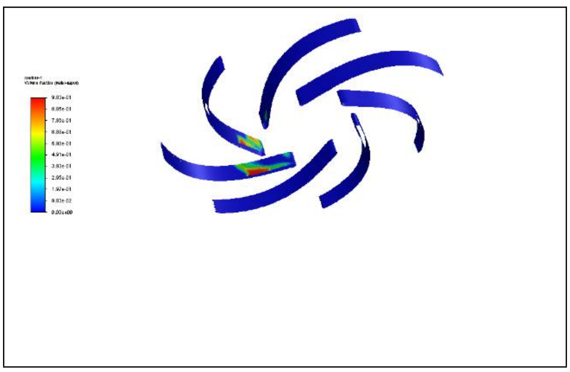


Figure 07 : Water vapor volume fraction for 2800 rpm and mass flow rate 0.667 kg/sec

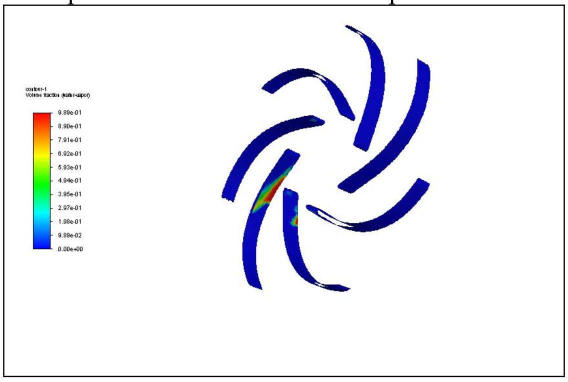


Figure 08 : Water vapor volume fraction for 2400 rpm and mass flow rate 1.0345 kg/sec

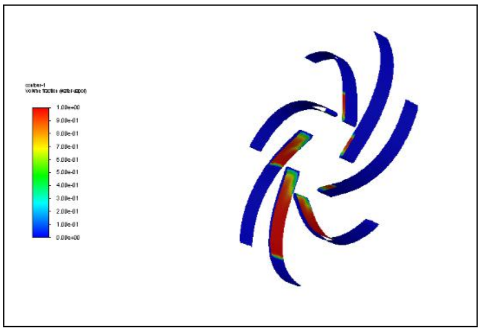


Figure 09 : Water vapor volume fraction for 2600 rpm and mass flow rate 1.0345 kg/sec

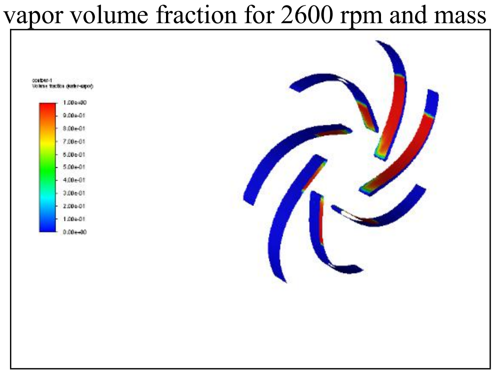
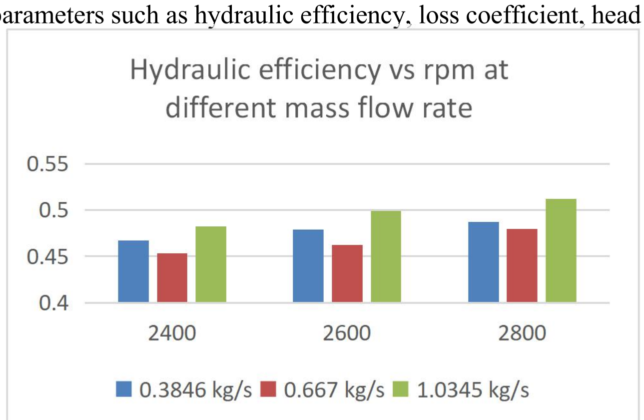


Figure 10 : Water vapor volume fraction for 2800 rpm and mass flow rate 1.0345 kg/sec

The above mentioned water vapor volume fraction contours are given to visualize the cavitaion effect in the centrifugal pump which may lead to severe erosion and operation breakdown of the pump blade. In above contours only that cases is shown or which cavitation occurred in the study. Rest of the case were cavitation free or might have negligible volume fraction of water vapor observed, are not shown here.

The following data has been observed after simulation and a comparative histogram has been generated that is shown below. All parameters such as hydraulic efficiency, loss coefficient, head coefficient etc. has been studied



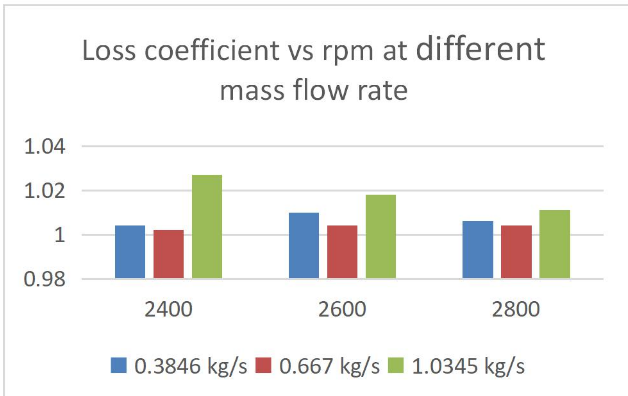


Figure 11 : Hydraulic efficiency and loss coefficient vs different mass flow rate

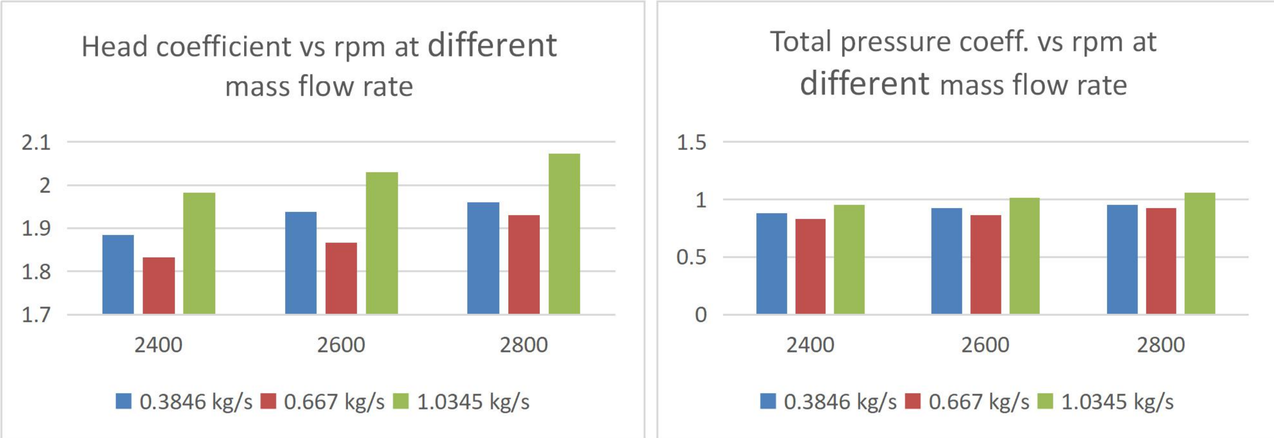


Figure 12 : Head coefficient and Total pressure coefficient vs different mass flow rate

VI. CONCLUSIONS

After analyzing the data obtained in the study following conclusion is drawn:

1. At constant mass flow rate of 1.0345 kg/s decreasing the rotational speed of impeller from 2800 rpm to 2400 rpm results in upsurge in hydraulic efficiency by 6.24%.
2. At constant mass flow rate of 0.667 kg/s decreasing the rotational speed of impeller from 2800 rpm to 2400 rpm results in upsurge in hydraulic efficiency by 3.13%.
3. At constant mass flow rate of 0.3846 kg/s decreasing the rotational speed of impeller from 2800 rpm to 2400 rpm results in upsurge in hydraulic efficiency by 4.17%.
4. The maximum efficiency in all cases occurs at 1.0345 kg/s mass flow rate and 2400 rpm.
5. At constant mass flow rate head coefficient upsurges with decrease in rpm which is 4.54% increase at 1.0345 kg/s, 3.62% at 0.667 kg/s and 4.25% at 0.3845 kg/s over the rpm ranging from 2400 to 2800 rpm.
6. At constant mass flow rate and varying the rpm total pressure coefficient is also found to increasing over the decrease in rpm and the maximum total pressure coefficient is found to be 1.06 at 1.0345 kg/s mass flow rate and 2400 rpm.
7. After analyzing the water vapor volume fraction contour it is found out that at fixed inlet and outlet pressure the probability of occurrence of cavitation in less as we decrease the mass flow rate. Also at fixed mass flow rate increasing the rpm upsurges the water vapor volume fraction which means chances of cavitation is more on increasing the rpm.
8. In the present study no cavitation is seen at 0.3846 kg/s mass flow rate for all rpm while a slight water vapour volume fraction is observed at 0.667 kg/s at 2800 rpm which increase at 1.0345 kg/s as we increase the rpm representing the worst condition for blade erosion and other sever breakdown of the operation of centrifugal pump in terms of cavitation.

REFERENCES

[1] Minggao Tan, Shouqi Yuan, Houlin Liu, Yong Wang and Kai Wang, (2009), Numerical Research on Performance Prediction for Centrifugal Pumps, Chinese Journal of Mechanical Engineering, Vol. 23, pp. 1-6.
[2] Karanth k. Vasudeva and Sharma N. Yagnesh, (2009), Numerical Analysis on the influence of Varying Number of Diffuser Vanes on Impeller-Diffuser Flow Interaction in a Centrifugal Fan, World Journal of Modeling And Simulation, Vol. 5, pp. 63-71.
[3] Bacharoudis E.C., Filios A.E., Mentoz M.D. and Margaris D.P., (2009), Parametric study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle, The Open Mechancial Engineering Journal, Vol. 2, pp. 75-83.
[4] Ozturk Adnan, Aydin Kadir, Sahin Besir and Pinarbasi Ali, (2009), Effect of Impeller- Diffuser Radial Gap Ratio in Centrifugal Pump, Journal of Scientific & Industrial Research, Vol. 72, pp. 203-213.
[5] Houlin Liu, Yong Wang, Shouqi Yuan, Minggao Tan and Kai Wang, (2010), Effects of Blade Number on Characteristics of Centrifugal Pumps, Chinese Journal of Mechanical Engineering, Vol. 23, pp. 1-6.
[6] Aman Abdulkadir, Kore Sileshi and Dribss, (2011), Flow Simulation and Performance Prediction of Centrifugal Pumps using CFD, Journal of EEA, Vol. 28, pp. 59-65.
[7] Jafarzadeh B., Hajari A,. Alishahi M.M. and Akabri M.H., (2011), Virtual simulatio of a Low-Specific-Speed High-Speed Centrifugal Pump, Applied Mathematical Modelling, Vol. 35, pp. 242-249.
[8] Yuan Jianping, Zhang Weijie, Jin Rong, Li Shujuan and Sunflow Wei, (2011), Flow Numerical Analysis within Auxiliary-Impellers of Centrifugal Pumps, International Conference on Information Science and Technology Nanjing, Jiangsu, China, pp. 1159-1163.
[9] Yang Sunsheng, Kong Fanyu and Chen Bin, (2011), Research on Pump Volute Design Method Using CFD, International Journal of Rotating machinery, Vol. 2011, pp. 1-7.
[10] Shojaeefard M.H., Tahani M., Ehghaghi M.B., Fallahian M.A. and M. Beglari,(2012), Numerical Study of the Effects of Some Geometric Characteristics of a Centrifugal Pump Impeller That Pumps a Viscous Fluid, Computer and Fluid, Vol. 60, pp. 61-70.

[11] Chakraborty Sujoy, Pandey K.M., Roy Bidesh, (2012), Numerical Analysis on Effects of Blade Number Variations on Performance of Centrifugal Pumps with Various Rotational Speeds, International Journal of Current Engineering and Technology, Vol. 2, pp. 143-152.
[12] Chakraborty Sujoy, Choudhari Kishan, Dutta Prasenjit, Debbarma Bishop, (2013), Prediction of Centrifugal Pumps with Variations of Blade Number, Journal of Scientific and Industrial Research, Vol. 72, pp. 373-378.
[13] Ozturk Adnan, Aydin Kadir, Sahin Besir and Pinarbasi Ali, (2013), Effect of Impeller Diffuser Radial Gap Ratio in a Centrifugal Pump, Journal of Scientific and Industrial Research, Vol. 68, pp. 203-213.
[14] Hussein, Mohammed Ali Mahmood, (2013), Effect of Rotational Speed Variation on the Static Pressure in the Centrifugal Pump, Journal of Mechanical and Civil Engineering, Vol. 8, pp. 83-94.