Experimental and CFD Analysis Of Centrifugal Pump Impeller- A Case Study

Mr. Jekim J. Damor*, Prof. Dilip S. Patel **,

Prof. Kamlesh H.Thakkar ***, Prof. Pragnesh K. Brahmbhatt ****

\(M.E CAD/CAM (Student) Mechanical Engineering, SPCE ,Visnagar, India

*** (Associate Professor, Mechanical Department, SPCE, Visnagar, Gujarat, India*

**** (Assistance Professor, Mechanical Department, SPCE, Visnagar, Gujarat, India,*

*****(Associate Professor, Mechanical Department, GEC, Modasa(38315),Gujarat, India*

Abstract

To analyze the centrifugal pump using the CFD techniques and predicting the performance of a mixed flow-type impeller of centrifugal Pump, in this paper, Experimental Investigations were conducted on centrifugal water pump with a 111 mm outlet impeller diameter, backward curved blades, nominal discharge of 4.00 lps and 12 m of head to assess the effect of various operating conditions like Head, Discharge, Power and Speed on the performance of the pump. Further the impeller is modelled using Solid works software and Computational Fluid Dynamics (CFD) analysis is carried out using ANSYS CFX software on the developed model of impeller to predict the performance virtually and to verify with the experimental result of the pump.

1. Introduction

Computational fluid dynamics (CFD) analysis is being increasingly applied in the design of centrifugal pumps. With the aid of the CFD approach, the complex internal flows in water pump impellers, which are not fully understood yet, can be well predicted, to speed up the pump design procedure. Thus, CFD is any important tool for pump designers. The use of CFD tools in turbo machinery industry is quite common today. Many tasks can numerically be solved much faster and cheaper than by means of experiments. Nevertheless the highly unsteady flow in turbo machinery raises the question of the most appropriate method for modelling the rotation of the impeller.

CFD analysis is very useful for predicting pump performance at various mass-flow rates. For designers, prediction of operating characteristics curve is most important. All theoretical methods for prediction of

efficiency merely give a value; but one is unable to determine the root cause for the poor performance. Due to the development of CFD code, one can get the efficiency value as well as observe actual. Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labour intensive, reducing time and, hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time. As a result of these factors, Computational Fluid Dynamics is now an established industrial design tool, helping to reduce design time scales and improve processes throughout the engineering world. CFD provides a cost-effective and accurate alternative to scale model testing with variations on the simulation being performed quickly offering obvious advantages From such literature, it was found that most previous research, especially research based on numerical approaches, had focused on the design or near-design state of pumps. Few efforts were made to study the off-design performance of pumps. Centrifugal pumps are widely used in many applications, so the pump system may be required to operate over a wide flow range in some special applications.

Numerical simulation of centrifugal pumps is not easy due to the usual CFD difficulties: turbulence, separation, boundary layer, etc. Although there are also specific problems: Complex geometry: a great number of cells is needed and, due to skewness, usually unstructured grids give better convergence than structured ones. Energy transfer is generated mainly by the centrifugal force in the impeller. A cascade simulation is not valid and these force source terms must be included in the equations of the moving zone.

CFD has proved to be a very useful tool in the analysis of these turbo machines, both in design and performance prediction.

The purpose of the present study is to show a numerical study of a centrifugal pump impeller taking into account the whole 3-D geometry and the unsteadiness of the flow. It has been done with the commercial software package CFX. Code uses the finite volume method and solves the k-ε equations with ability to handle unstructured grids, include relative reference frames and make unsteady calculations with moving meshes.

2 .Governing Equations

The incompressible flow through the rotating impeller is solved with a moving frame of reference with constant rotational speed.3-D incompressible Navier-Strokes equations with the rotational force term are solved to analyze the flow in centrifugal pump. Turbulence is modelled with k-ε turbulence model.

Mass Conversation Equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_j\right) = 0 \qquad .....(1)$$

In this equation U_j represented the three dimensional velocity vector components of the flow. If the flow is assumed steady, $\frac{\partial \rho}{\partial t} = 0$ and the equation reduced to

$$\frac{\partial U_j}{\partial x_j} = 0 \qquad .....(2)$$

Momentum Conversation Equation

The conversation equation for momentum ρU_i, can be formulated as

$$\frac{\partial}{\partial t}(\rho U_i) + \frac{\partial}{\partial x_i}\left(\rho U_i U_j\right) = -\frac{\partial P}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} + \rho f_i \qquad ...(3)$$

The three terms on right hand side of above equation represented the x_i components of all forces due to the pressure P, the viscous stress tensor $\tau_{ij,}$ stress tensor is given by

$$\tau_{ij} = -\mu_b \delta_{ij}\left(\frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial x_j} + \frac{\partial U_k}{\partial x_k}\right) - \mu\left(\frac{\partial U_I}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) \quad ...(4)$$

Where μ_b=2/3μ is the bulk viscosity, μ is the dynamic viscosity and δ_{ij} represented the kronecker delta (δ_{ij}=1 and i=j and δ_{ij}=0 for i≠j).

For flow in rotating frames of reference, the effects of the coriolis and centrifugal forces are modelled in the code. In this case

$$\vec{f_i} = 2\vec{\Omega} \times \vec{U} + \vec{\Omega} \times (\vec{\Omega} \times \vec{r} \qquad ...(5)$$

where vector notation has been used: x is cross product, Ω is the rotation velocity and $\vec{r}$ is the location vector.

Energy conversation equation

Besides mass and momentum energy is third fluid property for which a conversation equation in terms of the total enthalpy, H is given by

$$\frac{\partial}{\partial t}(\rho H) + \frac{\partial}{\partial x_j}\left(\rho U_j H\right) = \frac{\partial P}{\partial t}\left(U_i \tau_{ij} + \rho U_i f_i\right) \qquad ....(6)$$

where $H = h + \frac{1}{2} U_i U_i$,h=static enthalpy

If dissipation is small, neglecting pressure and dissipation terms

$$\frac{\partial}{\partial t}(\rho H) + \frac{\partial}{\partial x_j}\left(\rho U_j, H\right) = -\frac{\partial Q_j}{\partial x_j} + \rho U_i f_i \qquad ......(7)$$

In a rotating frame of reference, the enthalpy, i is advanced in place of the total enthalpy H.

The equation for I is given by

$$I = H - \frac{\omega^2 R^2}{2} \qquad ....(8)$$

Where w the rotation rate and R is local radius

3. Experimental Work

All this experiment work was done in LA GAJJAR. PVT. LTD, Ahmadabad.
Experimental setup is successfully developed and assembled without having any leakage problem. Experiments were carried out using water as working fluid.

Fig.1: Experimental Work

Operating frequency as 50Hz. Also setup was operated on different operating condition and effects were obtained. Analyst software used take reading in this experimental set up.
Experimental results at various operating conditions are shown in the following table.

Table 1: Experimental Results

Reading No	Mass Flow rate(kg/s)	RPM	Head (m)	Efficiency (%)
1	8.78	2842	7.55	33.20
2	6.83	2848	11.48	41.61
3	6.25	2858	12.48	43.64
4	5.30	2866	13.48	42.67
5	0.00	2918	16.03	0.00

Table 2: Design Specification of Impeller

Outer diameter of Impeller	111mm
Inlet diameter of Impeller	52 mm
Inlet blade angle	73.5˚
Outlet blade angle	56.5˚
Thickness of blade	3mm
Number of blades	7
Shaft diameter	14mm
Key	4 by 4mm

4. Modeling of Existing Centrifugal Pump Impeller

In order to study the effect of various parameter and for the ease of operation the existing pump component have been remodeled using the solidworks software

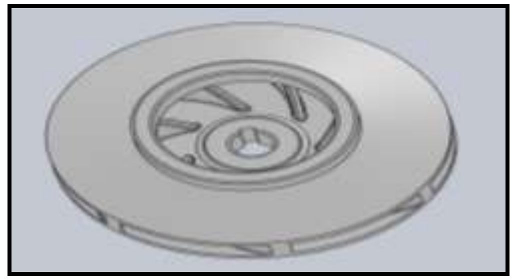

Fig.3 Modeling of centrifugal pump impeller

5. Meshing

The geometry and the mesh of a 7 bladed pump impeller domain generated using Ansys Workbench. An unstructured mesh with tetrahedral cells is also used for the zones of impeller and volute as shown in Figure.

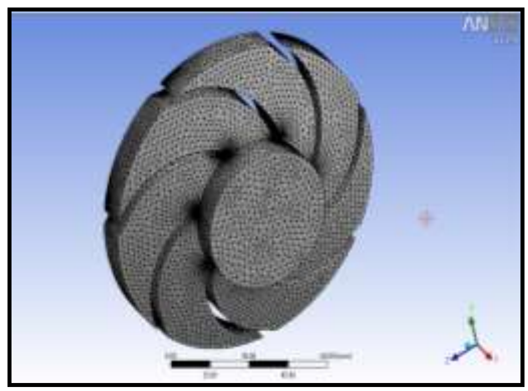

Fig.4 Meshed Model of Impeller Cavity

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.3 shows the mesh near the tongue region. A total of elements are generated for the impeller domain.

Table. 3: Mesh statistics

No. of Nodes: -	111637
No. of Elements:-	149456

6. Boundary conditions:

Centrifugal pump impeller domain is considered as rotating frame of reference with different rotational speed of 2842RPM, 2848RPM, 2858 rpm, 2866 rpm, 2918 rpm. The working fluid through the pump is water at 300k. k-ε turbulence model. Inlet static pressure and outlet different mass flow rate of 8.78 kg/s,6.83 kg/s, 6.25 kg/s,5.30 kg/s,0.00 kg/s are given as boundary conditions. Three dimensional incompressible N-S equations are solved with Ansys-CFX Solver.

7. CFD Analysis

CFD Analysis is carried out for various operating condition. Convergence criteria 1e-4 and 100 outer loop iteration.The results in form of Pressure and Velocity Contour are shown in the figures.

Case-1) Speed: 2842 rpm, Flow Rate: 8.78 lps

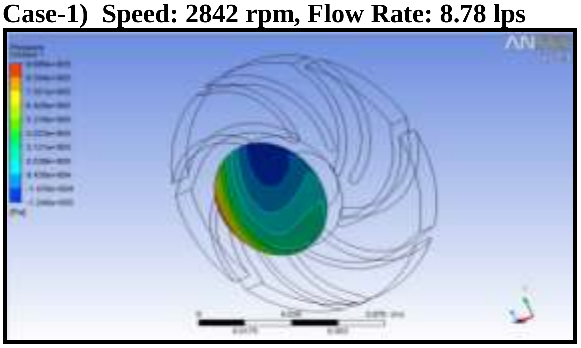

Fig.5: Inlet pressure contour

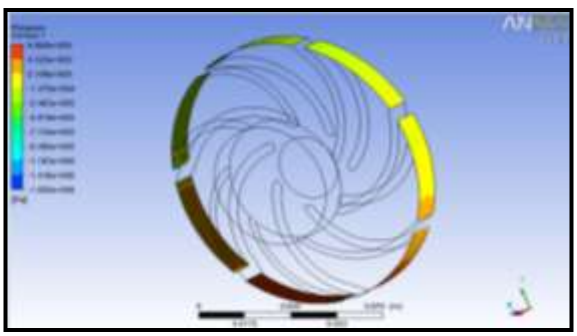

Fig.6: Outlet Pressure Contour

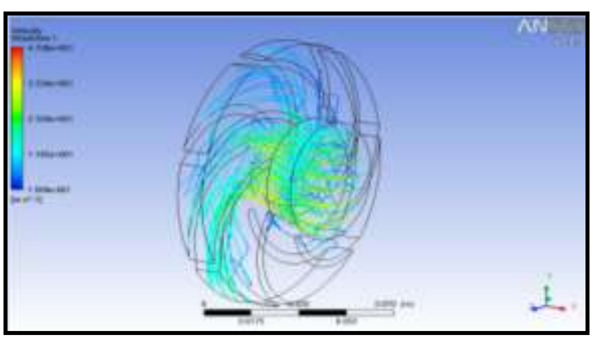

Fig.7:Velocity Stremline Contour

Case-2) Speed: 2848rpm, Flow Rate: 6.83 lps

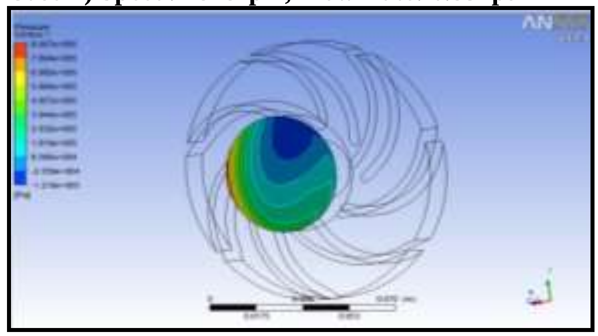

Fig.8:Inlet Pressure Contour

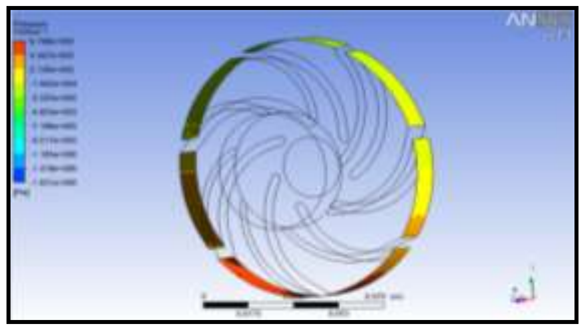

Fig.8 Outlet Pressure Contour

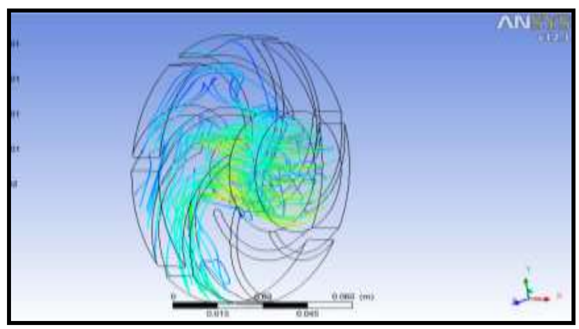

Fig.9:Steram Line Velocity Contour

Case-3) Speed: 2858rpm, Flow Rate: 6.25 lps

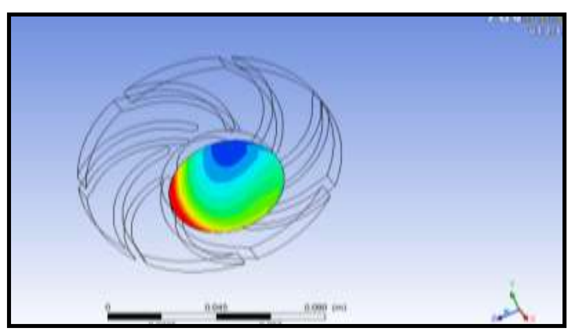

Fig.10:Inlet Pressure Contour

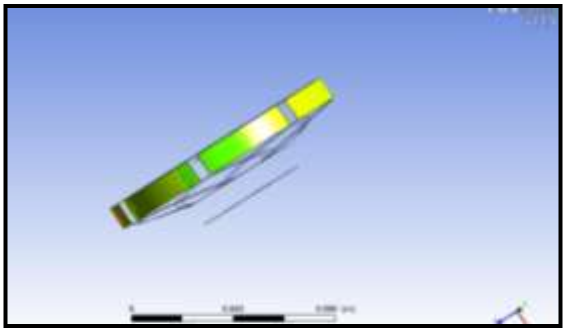

Fig.11:Outlet Pressure Contour

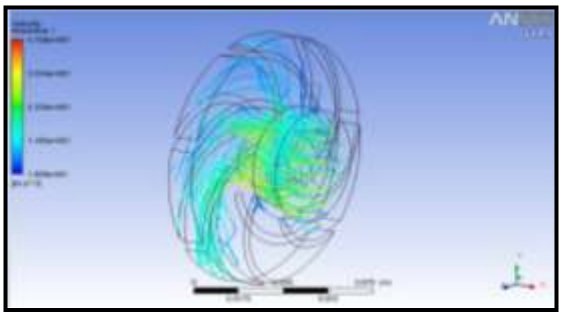

Fig.12:Sterammline Velcoity Contour

Case-4) Speed: 2866rpm, Flow Rate: 5.30lps

Fig.13:Inlet Pressure Contour

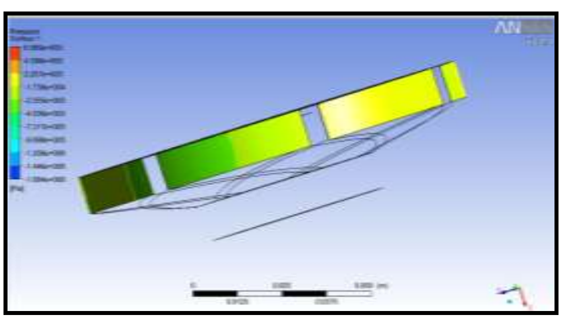

Fig.14:Outlet Pressure Contour

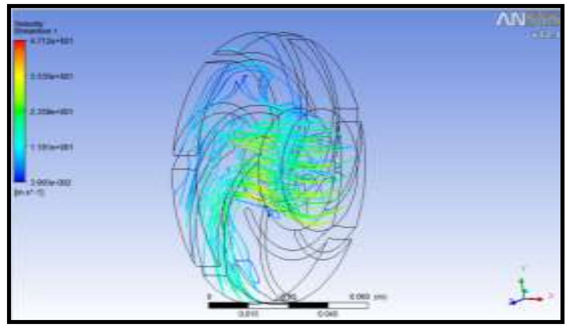

Fig.15:Steramline Velocity Contour

Case-5) Speed:2918rpm, Flow Rate: 0.00lps

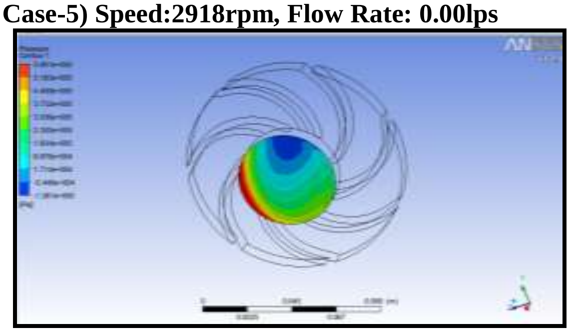

Fig.16:Inlet Pressure Contour

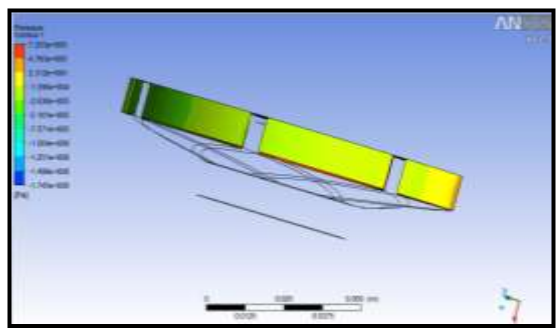

Fig.17: Outlet Pressure Contour

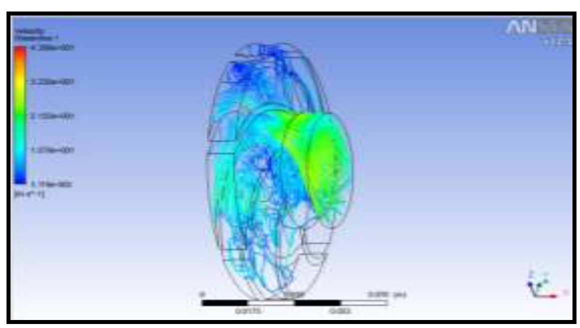

Fig.18: Streamline Velocity Contour

Results obtained after the CFD Analysis are following graphs.

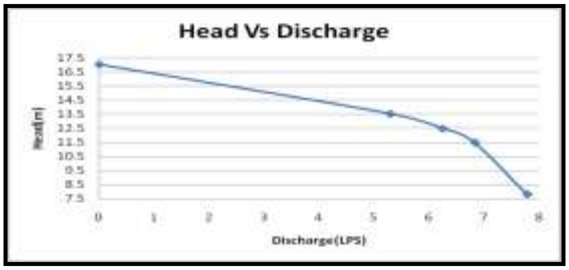

Fig.19:Graphically curve CFD Result Head vs Discharge

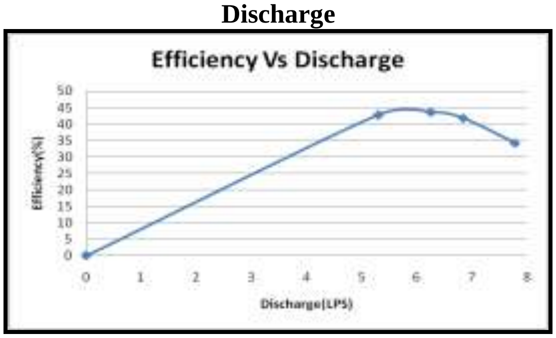

Fig.20: Graphically curve Efficiency vs Discharge

8. Result and Disccusion

Results obtained by Experimental work and the CFD Analysis are plotted on the same graph for the comparision.

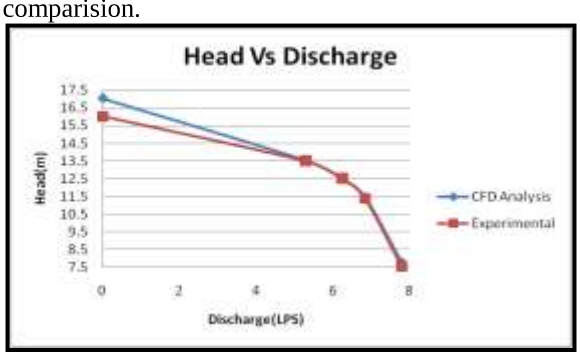

Fig.21:Charcteristics Head vs Discharge

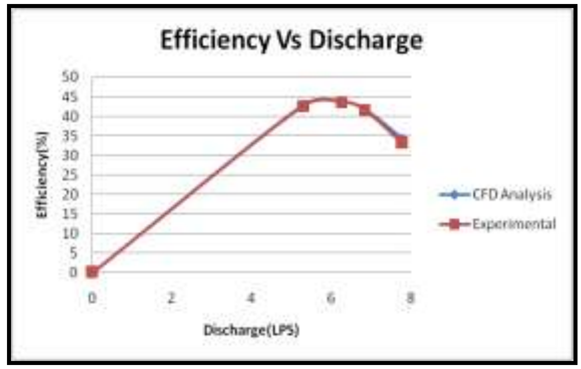

Fig.22: Characteristics Efficinecy vs Discharge

9. Conclusion

Experiments are carried out at various operating condition and same cases when analyzed using software the results obtained are almost satisfying. This proves the accuracy of CFD and Experimental approaches.

The best operating condition is flow rate 6.25 lps, head 12.49 m, efficiency 3.67% for the selected pump. It can be selected for the further analysis to improve the operating performance of the pump.

NOMENCLATURE

t Blade thickness
D Impeller diameter
H Pump head
Q Flow rate
η Hydraulic efficiency
ρ Density
α Inlet vane angle
β Outlet vane angle

References

[1] A. Manivannan "Computational fluid dynamics analysis of a mixed flow pump impeller" International journal of Engineering, science and Technology Vol. 2, No. 6, 2010, pp. 200-206

[2] Jose gonz'lez joauin fernandez eduardo blanco carlos santorlia "Numerical Simulation of the Dynamic Effects Due to Impeller-Volute Interaction in a Centrifugal Pump" http://www.asme.org354 Õ Vol. 124, JUNE 2002

[3] Perez J, Chiva S, Segala W, Morales R, Negrao C, Julia E, Hernandez LV "Performance Analysis Of Flow In A Impeller-Diffuser Centrifugal Pumps Using CFD :Simulation And Experimental Data

Comparisons" European Conference On Computational Fluid Dynamics Eccomas CFD 2010

[4] Michal Varcholaa, Peter Hlbocanb "Geometry Design of a Mixed Flow Pump Using Experimental Results of on Internal Impeller Flow" Procedia Engineering 39 (2012) 168 – 174

[5] Michal Varcholaa, Peter Hlbocanb "Prime Geometry Solution of a Centrifugal Impeller Within a3D Setting" Procedia Engineering 39 (2012) 197 – 203

[6] Maitelli, C. W. S. de P.1;, V. M. de F.; da Mata, "Simulation Of Flow In A Centrifugal Pump Of ESP Systems Using Computational Fluid Dynamics" Brazilian Journal Of Petroleum And Gas V. 4 N. 1 P. 001-009 2010 Issn 1982-0593

[7] P.Usha shri and C.Syamsundar "computational analysis on performance of centrifugal pump Impeller". Proceeding of the 37th National&4t,International conference on fluid mechanics and fluid power December 16-18,2010,IIT Madras,chenenai,India.FMFP10-TM-07

[8] Jose Gonzalez joaquin fernandez eduardo blanco carlos sntolaria "Numerical simulation of the dynamics effects due to impeller-volute interaction in a centrifugal pump".348/vol.124,june 2002,Transaction of the ASME.

[9] Suthep kaewnai, Manuspong Charmaoot and Somchai Wongwises "prediction performance of radial flow type impeller centrifugal pump using CFD". journal of mechanical science and Technology 23(2009) 1620-162