International journal of Advanced Scientific and Technical Research
Issue 1, Vol 1 October 2011
ISSN 2249-9954.

SIMULATION OF THREE DIMENSIONAL FLOWS IN HYDRAULIC PUMPS

D.K.Ramesha[1*]. G.PremaKumara[2], K.V.Sharma[3], Raghuprasad T.V[4],Madhusudan M[5].

[1] Associate Professor, [2]Asst Professor, [3]Professor and Head, [4,5] ME Scholar.

Thermal Science and Engineering,Department of Mechanical Engineering

University Visvesvaraya College of Engineering, Bangalore University,

K R Circle, Bangalore- 560 001,Karnataka, INDIA

[*]e-mail: rameshdknitk@gmail.com

ABSTRACT

Hydraulic efficiency of the centrifugal pump is mainly depends on mass flow rate, Head and RPM. Finding the best combination with the above said parameters and the best geometry of the pump is possible by using three dimensional CFD simulations. 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 behaviour. One can find the root cause for poor performance by using CFD Analysis of equipment. We have considered centrifugal pump for the three dimensional fluid flow analyses and validated the pump hydraulic efficiency with the experiment results. We observed the behaviour of the hydraulic efficiency with respect to the pump performance characteristics. Centrifugal pumps should be run at best efficiency point. Generally the radial force developed on the shaft will be high, if operates away from the BEP and it may leads to shaft damage. So it is important to find out best efficiency point.

Key words: *Best efficiency point, Head, Performance characteristics, Hydraulic efficiency*.

1. INTRODUCTION

The design of hydraulic turbo machines has reached the stage where improvements can only be achieved through a detailed understanding of the internal flow. The prediction of the flow in such equipment is very complicated due to the rotation and the curved three-dimensional shape of the impellers. Furthermore, the flow in turbo machines shows unsteady behaviour, especially at off-design conditions, as a result of interaction between impeller and pump casing. Considering these complexities, computer simulations will become increasingly important.

In this paper it is shown that the flow in hydraulic pumps of the radial type, operating at conditions not too far from design point, can be considered as an incompressible potential flow, where the influence of viscosity is restricted to thin boundary layers, wakes and mixing areas. A three-dimensional method for unsteady flow based on this model yields good results. The numerical method developed for solving unsteady potential flow is based on a fully three- dimensional finite-element method.

The capability of the method is demonstrated by analyses of the flow in several laboratory and industrial pumps. For these pumps, information on the velocity and pressure distribution as well as the performance characteristics is available from experiments. In general, computational results show a good agreement with experimentally obtained values. This and the short computing times required makes the proposed method very well suited as an analysis-tool within a pump development. By definition, a centrifugal pump is a machine. More specifically, it is machine that imparts energy to a fluid. This energy infusion can cause a liquid to flow rise to higher level or both.

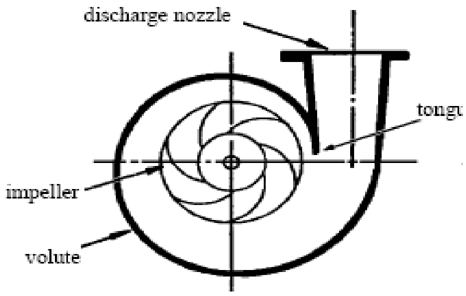

Figure1: Centrifugal pump and its parts

ISSN 2249-9954.

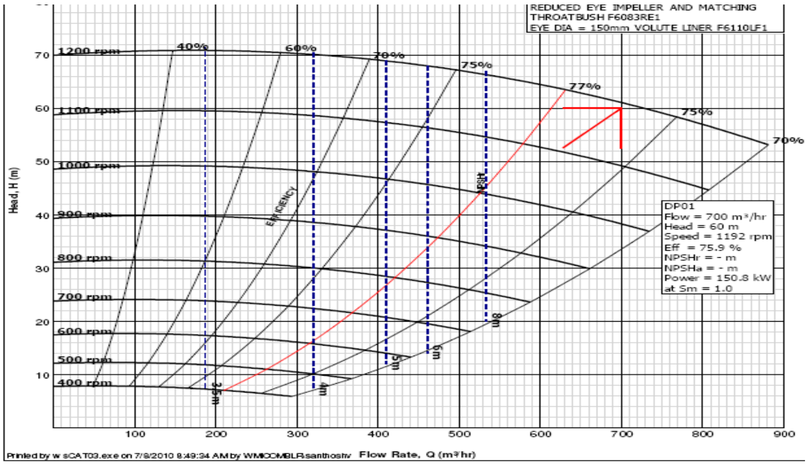

Figure2: Experimental data

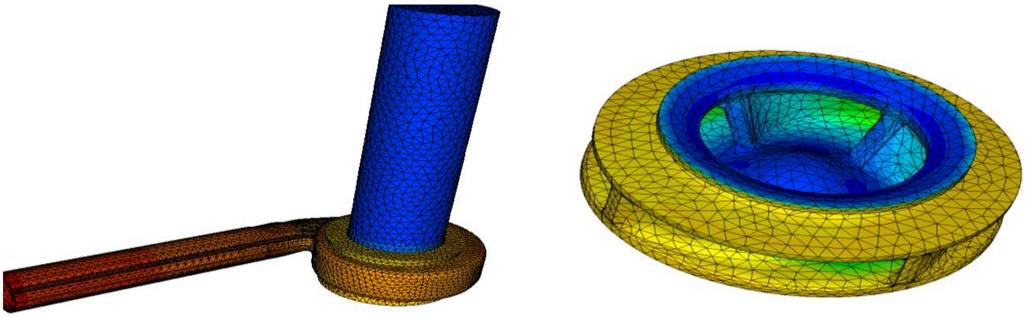

Figure 3: Zoomed View of Grid generation in the computational domain

2. BOUNDARY CONDITIONS

An inlet and out let pipe added to the geometry to maintain hydrodynamic ally fully developed region.

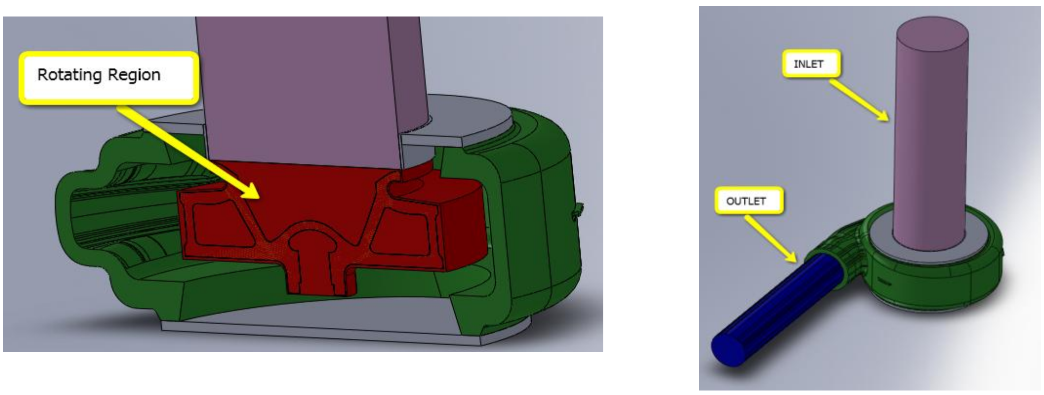

Figure 4: Boundary conditions.

The region of the flow in which the effects of the viscous shearing forces caused by fluid viscosity are felt is called the velocity boundary layer or just the boundary layer. The hypothetical boundary surface divides the flow in a pipe into two regions, the boundary layer region, in which the viscous effects and the velocity changes are significant, and the irrotational (core) flow region, in which the frictional effects are negligible and the velocity remains essential constant in the radial direction.The thickness of this boundary layer increases in the flow direction until the boundary layer reaches the pipe centre and thus fills the entire pipe, here pipes are suction and discharge pipes. The region from the pipe inlet to the point at which the boundary layer merges at the centreline is called hydrodynamic entrance region, and the length of this region is called hydrodynamic entry length. Flow in the entrance region is called hydrodynamically developing flow. The region beyond the entrance region in which the velocity profile is fully developed and remains unchanged is called the hydrodynamically fully developed region. The atmospheric pressure 101325 Pa is applied; at the out let of the pump mass flow rate Q applied, rotating region added around the impeller and rotation RPM is applied to the rotating region.

3. RESULTS AND DISCUSSION

To calculate the hydraulic efficiency of a pump at duty points of experimental data, the simulations carried out for varied mass flow rate and RPM of the impeller.

Hydraulic efficiency calculated from the pressure head, hydraulic torque and velocity of the impeller obtained from the simulation.

$$\eta = \frac{\overset{[kg/m^3]}{\rho}\ \overset{[m/s^2]}{g}\ \overset{[m^3/s]}{Q}\ \overset{[mH_2O]}{\Delta P}}{\underset{[Nm]}{T}\ \underset{[rad/s]}{n}}$$

Simulation carried out at all duty points, the hydraulic efficiency at each duty points are within 5% variation which is acceptable.

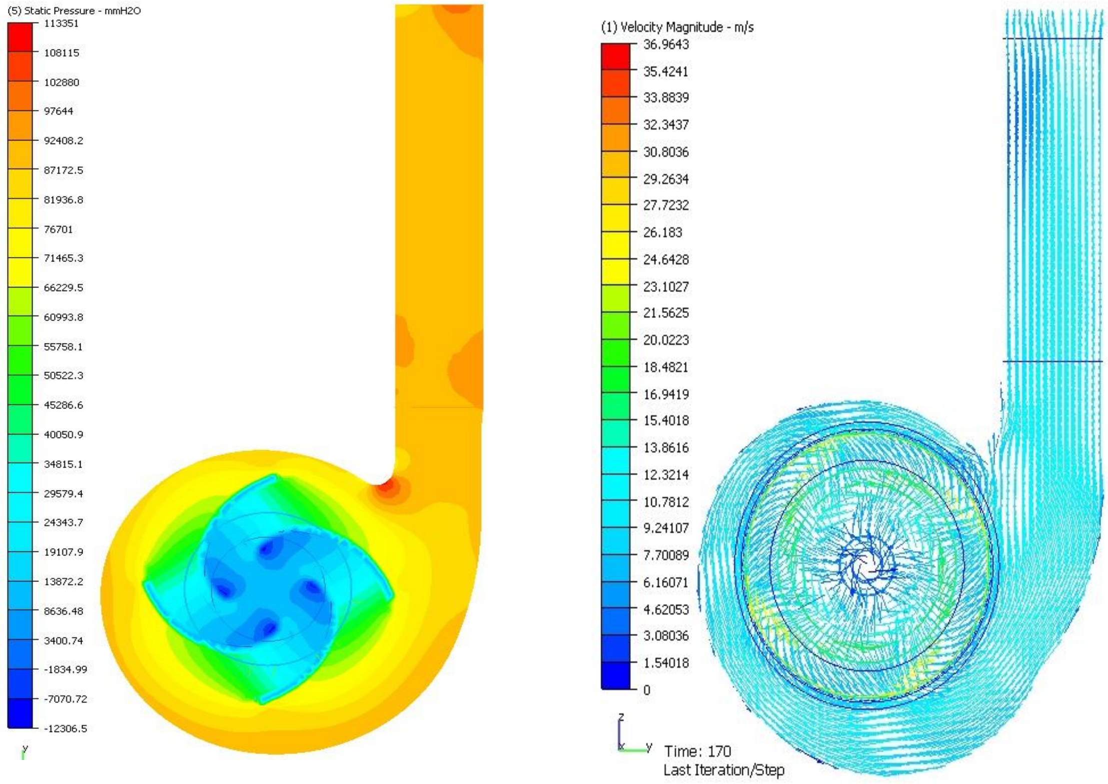

Figure 5: Pressure and velocity distributions.

Table1: Comparison between simulation and experimental results.

Final Data					
Duty points	Q	H	η Experimental	η Simulation	% Error
1	0.00	70.00	0.4	0.38	2
2	160.00	74.00	0.4	0.38	2
3	280.00	71.00	0.6	0.57	3
4	390.00	68.00	0.7	0.67	3
5	495.00	67.00	0.75	0.73	2
6	640.00	62.00	0.77	0.74	3
7	700.00	60.00	0.76	0.73	3

8	770.00	58.00	0.75	0.72	3
9	880.00	54.00	0.7	0.69	1

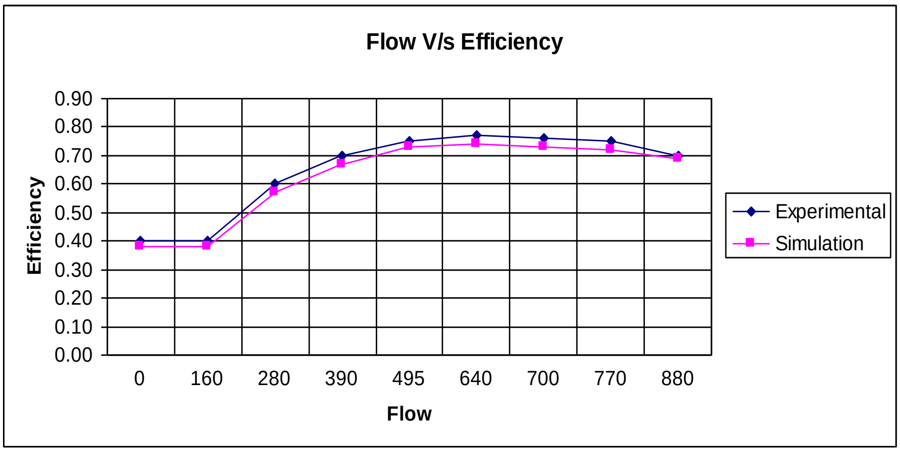

Fig 6: Comparison between simulation and experimental results.

4. CONCLUSIONS

Hydraulic efficiency of the centrifugal pump is mainly depends on mass flow rate, Head and RPM. Finding the best combination with the above said parameters and the best geometry of the pump is possible by using three dimensional CFD simulations.

- Pressure distribution in the pump studied for the varied mass flow rate, and RPM and calculated hydraulic torque.
- Compared the simulation results with the experimental results, the % error is within the acceptable range.
- The experimental and simulated value for the static pressure distribution at the impeller blade is reasonable. Simulated circumferential and radial velocity distributions across the impeller passage near the trailing edge are good agreement with experimental results.
- The simulated radial velocities are too small, possibly caused by boundary layer separation and tip clearance effect.
- At flow rates near best efficiency point, the simulated head characteristic agrees well with measurements. Standard models for boundary layer dissipation and wake mixing are used to quantify the viscous losses.
- At high flow rate, the head is over predicted, possibly because of boundary layer

separation in the volute.

- Computed circumferential velocities in the volute are in good agreement in regions not too close to the tongue.
- Best efficiency point calculated, the pump should run at BEP to avoid radial forces development on pump shaft.

5. REFERENCES

1. Aarts, R., 1997, "The computation of fluid-induced forces on centrifugal impellers rotating and whirling in a volute casing", Proc. Third international symposium on pumping machinery, ASME FED Summer Meeting, Vancouver, Canada.
2. Adkins, D.R., Brennen, C.E., 1988, "Analyses of Hydrodynamic Radial Forces on Centrifugal Pump Impellers," J. Fluids Engng., vol. 110, pp. 20-28.
3. Chamieh, D.S., Acosta, A.J., Brennen, C.E., Caughey, T.K., Franz, R., 1982, "Experimental Measurements of Hydraulic Stiffness Matrices for a Centrifugal PumpImpeller," Proc. of the 2nd NASA/ARD workshop on rotordynamic instability problems in high performance turbomachinery, NASA CP-2250.
4. Combès, J.F., Rieutord, E., 1992, "Numerical and experimental analysis of the flow in a centrifugal pump at nominal and partial flow rate," ASME paper 92-GT-284.
5. Duineveld, P.N., 1997, "LDA metingen van de relatieve snelheid in een mixed-flow waaier," M.Sc. Thesis (in Dutch), University of Twente.
6. Lee,J. and Lee, SJ, (2000), " The effect of nozzle configuration on stagnation region heat transfer enhancement of axisymmetric jet impingment", Int.J. Heat Mass Transfer, Vol. 43 No. 18, pp 3497-509.