Indian J.Sci.Res. 17(2): 228-231, 2017 ISSN: 0976-2876 (Print)
ISSN: 2250-0138(Online)

DESIGN & PERFORMANCE ANALYSIS OF DOUBLE SHROUDED CENTRIFUGAL PUMP'S IMPELLER USING CFD

[1]Dr. Rajiv Kaul
[1]Associate Professor, College of Military Engineering, Pune-31.

Abstract-Centrifugal Pump technology involves a wide spectrum of flow phenomenons and various methods of Impellers design, blade angles, fabrication and degree of Impellers surface finishing, which has a profound impact on its performance.The design of a Centrifugal Pump's Impellerdemands a detailed understanding of the internalflow during design and operatingconditions. The current investigation is aimed to simulate the complex internal water flow in a Centrifugal Pump's Impeller (6-blades, double-shrouded) and performance analysis by using a (3-D) Navier-Stokes equation with various turbulence models. The numerical solution of the deiscretizedincompressible Navier-Stokes equations over a hybrid grid (3-D model of flow domain) is accomplished with CFD package Ansys-Software. The aim of this study is to understand the mechanism and the various design aspects of the double shrouded Impeller for optimum performance.

***Key words*—**Impeller, Turbulence models, Grid, 3D Model, Navier-Stokes equaions

I. Introduction

The design methods for Centrifugal Pump have passed a long process of development.Flow analysis of Centrifugal Pump is often a challenging task as it requires critical analysis of highly complex flow whichis turbulent and three dimensional in nature and having rapidly changing curvature of flow passage.It is very crucial to understand the flowbehavior and variation in flow parameters in order topredict and analysis of performance on account of rotation and 3-D curved shaped of the impeller.Also flow through the Centrifugal pump is very complex mainly due to the rotation imposed by the impeller and its interaction with the volute casing particularly in double shrouded Impeller.Withtheaid of CFD, the complex internal flows in water pump impellers can be well predicted, thus facilitating the design ofpumps.However, a sensitive analysis is based on grid quality and turbulence model.

Nevertheless the highly unsteady flow in Pump raises the question of the most appropriate method for modelling the rotation of the impeller.

Many experimental works have been conducted to explore the details of inside fluid flow and pressure rise in the Centrifugal pump. Unfortunately, due to geometric complexity and experimental limitations, it is very difficult to capture reliable and accurate data on flow over blades profile particularly at blades exit tip. The blade number and its design are important design parameters of pumps, which affects the characteristics of pump heavily.

II. Analysis of Impeller's Design

The impeller (double shrouded) analysis for the radial flow Centrifugal pump was based on the design details. Speed 2500-3200 rpm, flow rate 1.5-4.5 lit/s, head developed 10-16m. The dimension of the impeller:- impeller diameter 134mm, total number of blades 6, blade angles at inlet and outlet 18^0, 30^0. The blade thickness in the middle is 4.2mm. The blade exit tips radius, 1.0 and 1.5mm. The analysis starts with 3-D modelling of the Impeller, then mesh generation (hybrid) and refinement of domain specifically where velocity and pressure gradients are high. Governing equations for flow through the Pump, are 3-D incompressible N-S equations and mass conservation equations.The simulation and analysis include flow fields in rotating impeller and stationary parts and is completed in a single rotating reference frame. Governing N-S equations

$$\rho\frac{Du}{Dt} = -\frac{\partial p}{\partial x} + div(\mu . grad\ u) + S_{MX}$$
$$\rho\frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + div(\mu . grad\ v) + S_{My}$$
$$\rho\frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + div(\mu . grad\ w) + S_{Mz}$$

& continuity equation $\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0$

Although all the boundary conditions are time invariant, the flow field in each of the blade passages can change with time as they rotate from one inlet sector to the next when there is unequal flow.

A. ***Computational model*** – The Computational results presented in this paper were all obtained using a general purpose CFD code--AnsysFluent (Software). This is a finite volume, unsteady N-S solver. In order to model reality as closely as possible, a full 3D model was created encompassing a full domain from the suction line (eye) to delivery (line). This means that the measured inlet conditions could be applied directly to the model. Mesh generation (67323 elements) was done using structured/unstructured tetrahedral grid for each component (half region due to symmetry). The mesh near

[1]Corresponding Author

the boundary wall is refined to catch the boundary layer effects.

This approach reduces the number of iterations required to obtain convergence and no pressure correction term is required to retain mass conversion, leading to a more robust and accurate solver. It is intended tosimulate the flow through the impeller of Centrifugal pump using finite-volume method along with agrid system for the solution of the discretized governing N-S equations. The CFD technique was applied to predict theflow patterns, pressure distribution and velocityprofile for various Impeller's deign.

III. Investigations on Interacting Components

The relative movement between impeller and volute generates an unsteady interaction which affects not only the overallPump performance but is also responsible for pressure fluctuations. Pressure fluctuations interact with the volute casing andgive rise to dynamic effects (mainly unsteady forces) over the mechanical parts, which are one of the most importantsources of vibration and hydraulic noise. Numerical simulation could capture the dynamic and unsteady flow effects inside a Centrifugal Pump due to Impeller-volute interaction. Viscous N-Sequations along with rotating mesh technique were applied to consider the impeller-volute interaction. Numerical prediction showed the presence of a spatial fluctuation pattern at theblade passing frequency as a function of the flow rate.

Design Parameters:- The turbulence was simulated withShear Stress Transport Model (SST model). The convergence criterion was 10^{-5}. All the wall surface roughness within the control volume was set to 100µm.

URANS equations together with two equation SST turbulence model were found to be appropriate to get a reasonable estimation of the general performance of the Centrifugal Pump, from an engineering point of view.

Denomination	**Value**
Suction pipe diameter	D_s =50mm
Impeller diameters (Aluminium)	D_1 = 58mm, D_2 = 134mm
Impeller widths	b_1=2mm, b_2= 3.5mm, h=7-8mm
Blade angles	**β_1 = 18 °, β_2 =30°**
Number of blades, Head	z = 6, 10-- 15 m
Flow rate in best efficiency point,	Q = 1.5 - 4.5 lit/s, N = 2500 to 3200 rpm, vel-- 1.58--3.38 m/s
Specific speed ($N_s = N \cdot Q^{1/2}/H^{3/4}$)	N_s = 650-1000rpm

The results of the k-ε are much less sensitive to the (arbitrary) assumed values in the free stream, but its near wall performance is unsatisfactory for boundary layers with adverse pressure gradients. This led to use (i) a transformation of the k-ε model into a k-ω model in the near–wall region (ii) the standard k-ε model in the fully turbulent region far from the wall. SST is the combination of both.

Fig 1.Double Shrouded Impeller

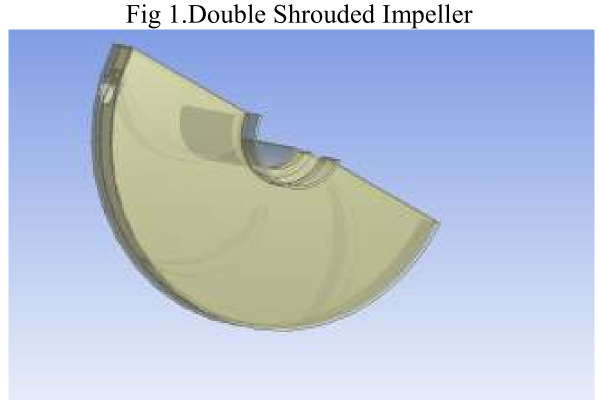


Fig 2.Double Shrouded Impeller

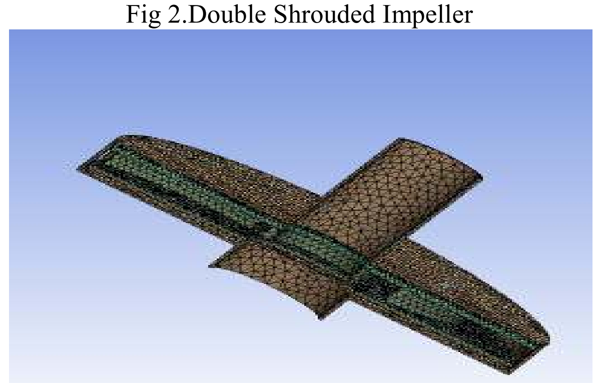


Fig. 3 Meshed Impeller

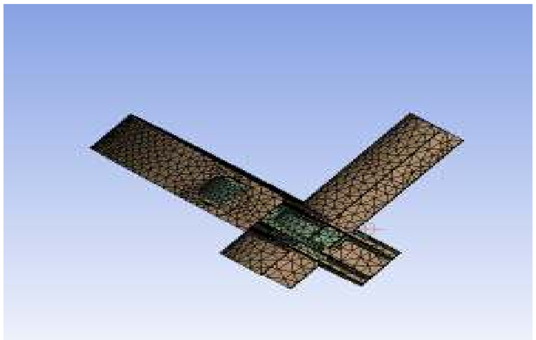


Fig. 4 Meshed Impeller

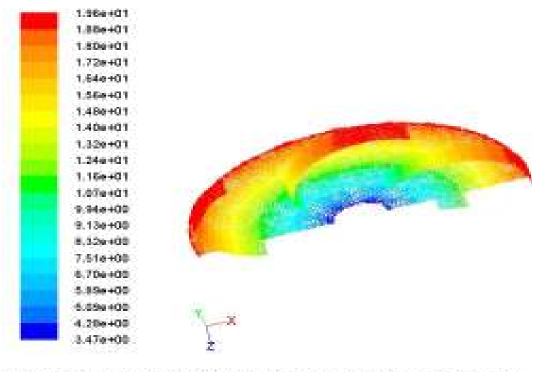


Fig. 5 Velocity Contour

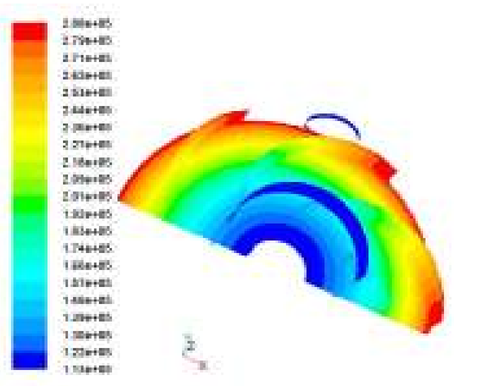
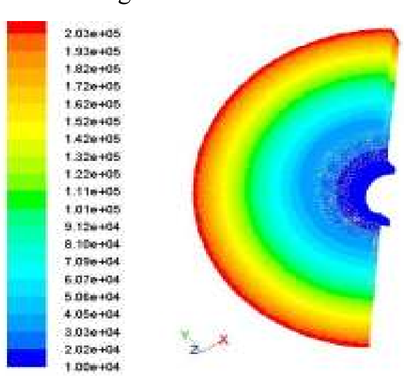


Fig. 6 Pressure Contour



IV. Conclusion

Blade is a rotary hydraulic part of pump which performs function of gathering high velocity fluid, transferring velocity energy into pressure energy. With double shrouded impeller, maximum velocity and pressure found over the blade profile, were of the order of 19.6m/sand 2.03×10^5 Pa whereas in single shrouded, they were, 37.1m/s and $1.72 \mathrm{X} 10^5$ Pa. It shows that with double shrouded impeller, tendency to impart the kinetic energy increases as the water glides over the blades profile as well as the corresponding increase in pressure head. The mechanical,recirculation,hydraulic and otherfriction losses are also less as the change in velocity and pressure occurs slowly during the journey of water over the blades profile as chances of splashing of water almost becomes zero.

At normal operating conditions, with double shrouded impeller**,** the velocity wasincreased by more than 8-10% but the hydraulic efficiency was increased by 2-5%. The corresponding increase in pressure head is of the order of 12-15%. However, at high flow rates and speeds, the increase is (up to 8%)for pressure head. The impeller geometry was represented by a number of controllable design variables,providing the capability of modifying the impeller shape and testing different configurations. The results of such parametric studies showed that, a remarkable gain in hydraulic efficiency may be achieved by optimizing the impeller geometry. The pressure contours show a continuous pressurerise from leading edge to trailing edge of theimpeller due to the dynamic head developed by therotating pump impeller. It is observed that totalpressure on pressure side of the blade is more thanthat of suction side. The total pressure patterns are varying along thespan of the impeller. Low total pressures areobserved near hub of the impeller. With increase inspan, total pressures are increasing because of highdynamic head at tip of the blade. A low totalpressure and high velocity is observed near theleading edge on suction side of the bladebecause of the vane thickness. It is to be noted that in double shrouded Impeller rise in weight is a negative factor which in turn, increases the input energy, hence rise in efficiency is insignificant despite rise in developed pressure head.

The predicted results are presented in terms of pressure profiles and velocity vectors.

References

[1] Sunsheng Yang, Fanyu Kong, Bin Chen,2011. Research on Pump Volute Design Method Using CFD, International Journal of Rotating Machinery, China, Vol 2011, [2]Keck, H., Weiss, T., Michler, W., Sick, M., 2007.Recent developments in the dynamic analysis of water turbines,Proceedings of the 2ndIAHRInternational Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic

Machinery and Systems, Timisoara, Romania, pp. 9-20.

[3] Pascoa, J., Mendes, A., Gato, L., 2009.A Fast Iterative Inverse Method for turbo-machinery Blade Design.Mechanics Research Communications36(5),p. 537.

[4] Article ID 137860,Keck, H., Sick, M., 2008. Numerical Flow Simulation in Hydraulic Turbo-Machines. ActaMech 201, p. 211.

[5] Kaupert, K., Holbein, P., Staubli, T., 1996. A First Analysis of Flow Field Hysteresis in a Pump Impeller, Journal ofFluids Engineering 118, p. 685.

[6] Potts, I., Newton, T., 1998. Use of commercial CFD pack-age to predict shut-off behavior of model centrifugal pump: an appraisal,IMechE Seminar.

[7] Sun, J., Tsukamoto, H. 2001. Off-Design Performance Prediction for Diffuser Pumps, Journal of Power and Energy, Proceedings ofI. Mech. E A215,p. 191.

[8] V Jain, R N Patel, V J Lakhera, S R Shaha, CFD for Centrifugal pumps: a review of the state-of-the art, Procedia Engineering 51 (2013) 715 – 720 1877-7058 © 2013 The Authors. Published by Elsevier Ltd.Selection and peer-review under responsibility of Institute of Technology, Nirma University, Ahmedabad.doi: 10.1016/j.proeng.2013.01.102 Chemical, Civil and Mechanical Engineering Tracks of 3rdNirma University International Conference (NUiCONE 2012)

[9] Tsui, Y., Lu, S., 2008.Evaluation of Performance of a ValvelessMicro-pump by CFD and Lumped System Analyses, Sensors and Actuators148, p. 138.

[10] http://www.cfdreview.com