Design and Analysis of Centrifugal Pump Impeller using CFD Analysis

Kaviarasan .R[1], Nagarajan. S[2], Prasanth. E[3]
[1, 2, 3] UG Scholar,
Department of Mechanical engineering
SNS college of Technology,
Coimbatore 641035,

Dr. Subramanian. M[4]
[4]Professor,
Department of Mechanical engineering
SNS college of Technology,
Coimbatore 641035,

***Abstract:* - Design optimization of a backward-curved blades centrifugal pump has to improve hydraulic performance of the pump using three-dimensional modelling and CFD analysis. This project investigates the study of complex internal flows in centrifugal pump impellers with the aid of Computational Fluid Dynamics software (CFD) thus facilitating the design of pumps. The pump specifications considered for investigation are discharge and speed. These specifications have been varied to perform a comparative study of these pump impellers .To enhance the performance of the centrifugal pump through design modification of impeller. The impeller is modelled in Pro engineering software and CFD analysis is done using fluid flow simulation package. CFD analysis enables to predict the performance of the pump and a comparative analysis is made for the entire control volume by varying meshing. Thus the valid results regarding the velocity distribution and pressure distributions are to predict and the performance of the centrifugal pump has to increase the hydraulic efficiency. A number of geometric variables are introduced for the parameterization of the impeller. Geometry allowing also for easy design modifications. The computations for the steady flow field in a particular impeller are presented and the results are analyzed with the aid of 3-dimensional graphs.**

1 INTRODUCTION:

Centrifugal pumps are prevalent for many different applications in the industrial or other sectors. Nevertheless, 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 hydrodynamic design for many different pump types. The numerical simulation can provide quite accurate information on the fluid behaviour in the machine, and thus helps the engineer to obtain a thorough performance evaluation of a particular design. However, the challenge of improving the hydraulic efficiency requires the inverse design process, in which a significant number of alternative designs must be evaluated. Despite the great progress in the latest years, even CFD analysis remains rather expensive for the industry, and the need for faster mesh generators and solvers is imperative.

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. An alternative practice in complex domains is the use of Cartesian grids that need a much reduced construction effort. However such grids cannot be everywhere body-fitted, and for this reason various numerical techniques have been developed to improve the accuracy in these regions [5-6]. A Cartesian mesh approach is also followed in the present work, where an advanced numerical technique is incorporated in order to eliminate the grid generation cost and to represent with adequate accuracy the complex geometry of a centrifugal pump impeller. The latter is parameterized using a reduced number of controlling geometric variables, facilitating the investigation of their individual or combined effects on the flow and the impeller performance. The results of the predicted flow field in the pump impeller illustrated in Fig. 1, as well as the numerically reproduced performance curves of the impeller are then presented and discussed.

1.1 Selection of pump for performance enhancement

The design and performance analysis of centrifugal pump are chosen for the project work, because it is the most useful mechanical rotor dynamic machine in fluid works which widely used in domestic, irrigation, industry, large plants and river water pumping system. These pumps are used at the place where the requirements of head and discharge are moderate. The performance of centrifugal pumps drops sharply during the pumping of viscous fluids. Changing some geometric characteristics of the impeller in these types of pumps improve their performance. In this investigation, the 3-D flow in centrifugal pump along with the volute has been numerically simulated. Experimental investigations are generally carried out on pumps which are expensive, time consuming and limited to some extent. To reduce the number of experimental works, virtual analysis can be carried out on different pump models with the use of CFD packages and pump performance can be predicted.

1.2 Functions of cfd analysis

Computational fluid dynamics (CFD) uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow (the Navier- strokes and allied equations), for predefined geometries and boundary conditions. The result is a wealth of the predictions for flow velocity, temperature, density, and chemical concentrations for any region where flow occurs. CFD analysis begins with a mathematical model of a physical problem, conservation of matter, momentum, and energy must be satisfied throughout the region of interest. Fluid properties and modelled empirically. Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, in viscid, two dimensional). Provide appropriate initial and boundary conditions for the problem. CFD applies numerical methods (called discretization) to develop approximations of the governing equations of fluid mechanical in the fluid region of interest. The solution is post processed to extract quantities of interest (e.g. lift, drag, torque, heat transfer, separation and pressure loss).

2 DESCRIPTION

2.1 Solid Modeling

CFD technique is used for the flow rate and pressure rate analysis of the centrifugal pump impeller through the ANSYS analysis software for which the 3-D model of the centrifugal impeller was made in the SOLID WORKS modeling software and then the impeller model made was converted into IGES format so that it can be supported by the ANSYS software and then model is opened in ANSYS fluid dynamics.

2.2 Geometry parameterization

The geometry of the particular radial flow impeller examined here (Fig. 1) corresponds to a model impeller constructed in the Lab, and can be represented using a relatively small number of parameters; most of them are shown in Fig. 2 and their values are given in Table 1. The rotation speed and the main impeller dimensions, namely the exit diameter and width $D2$ and $b2$, as well as the blade inlet and exit angles, $\beta1$ and $\beta2$, respectively, determine the nominal head and volume flow rate of the impeller. The computational domain is extended to a certain radial distance beyond the impeller outlet to prevent any backward influence of the free vortex conditions set at the exit boundary. The blades are constructed as circular arcs, and they have constant width and both edges rounded, allowing for both pump and turbine operation mode of the impeller. The rest parameters constitute the free design variables that can be modified in order to improve the performance and hydraulic efficiency of the impeller for this particular nominal operation
point.

2.3 Meshing of impeller

The meshing is done on the model .In order to get a better result, locally finer meshing is applied to the region which is suspected to have the highest pressure rate.

2.4 Design specification

Conventional impeller specification

Inlet Diameter (d1) = 74 mm
Outer Diameter (d2) = 200 mm
Specific speed (N) = 980 rpm
Number of vanes (z) = 6 (5 mm thick)
Breadth of Impeller (B) = 25 to10 mm (converging from inlet to outlet)
Inlet blade angle (ß1) = 20 degrees
Exit blade angle (ß2) = 24 degrees

Redesigned impeller specification

Inlet Diameter (d1) = 75 mm
Outer Diameter (d2) = 230 mm
Specific speed (N) = 1500 rpm
Number of vanes (z) = 6 (4 mm thick)
Breadth of Impeller (B) = 20.5 to 8 mm (converging from inlet to outlet)
Inlet blade angle (ß1) = 16 degrees
Exit blade angle (ß2) = 23.5 degrees

3 Design

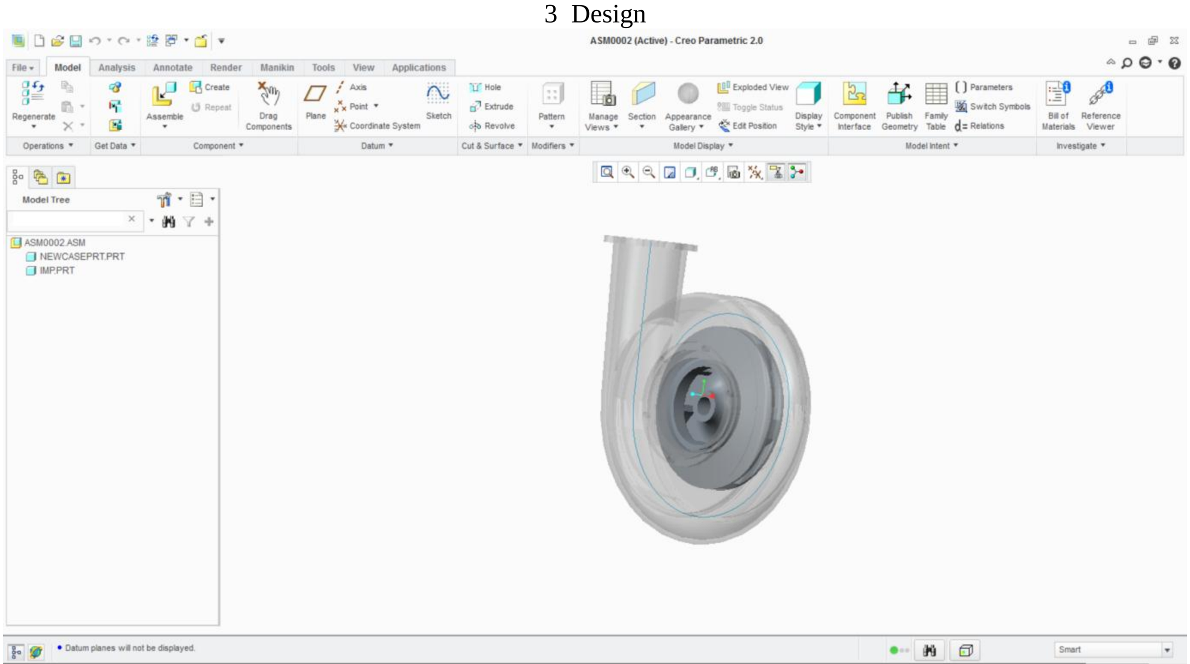


Fig 3.1 Impeller with case

Fig 3.2 Impeller 3D view

4 Analysis:

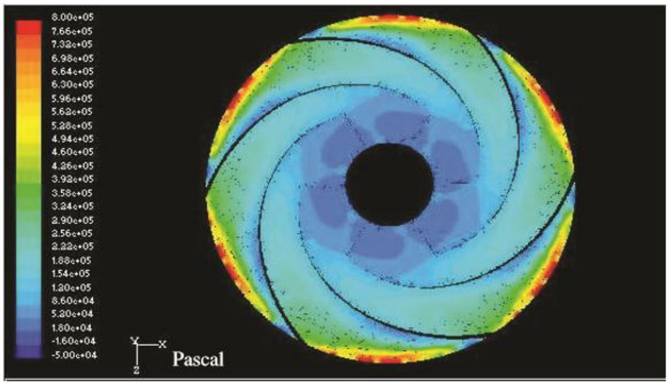


Fig 4.1 Analysis of conventionally designed model

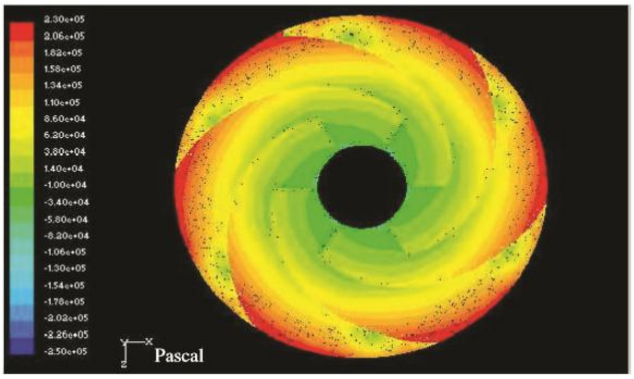


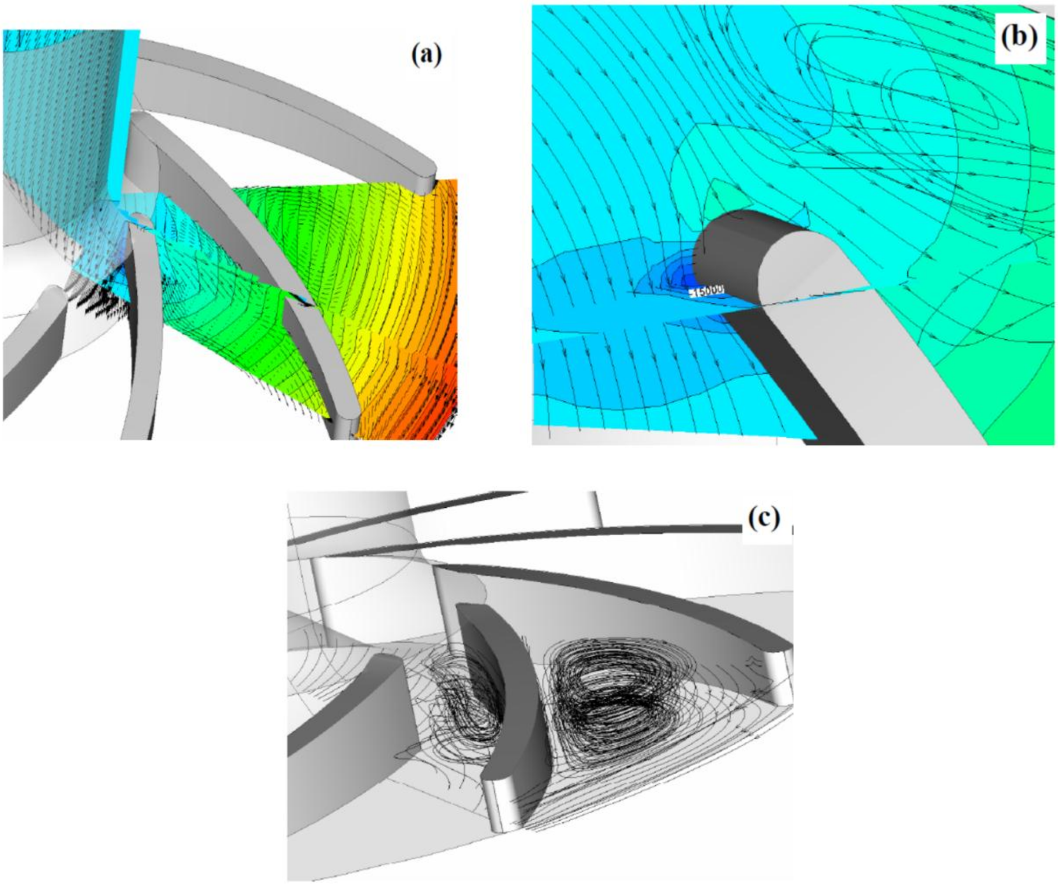
Fig 4.2 Analysis of conventionally designed model

5 RESULTS:

5.1 Flow analysis

At first the numerical model is applied to calculate the flow field developed in the standard impeller for various load conditions and a constant rotation speed of 1500 rpm. The ‘nominal’ volume flow rate is taken 31 m3/h to comply with the corresponding laboratory model pump design conditions.

(Fig. 5a) shows the resulting contours of pressure and the velocity vectors at two grid levels that cross the impeller normal to- and through its axis of rotation. Increased flow velocity can be observed at the blade inlet due to the blockage of the flow, whereas on the contrary the pressure is reduced. Further downstream the pressure contours become smooth between the blades and its value increases continuously towards the exit of the computational domain.



The minimum pressure appears, as expected, at the suction side and near the leading edge of the blade (Fig. 5b). The flow seems to enter almost parallel to the blade (Fig. 5b) and the streamlines follow a regular pathway between the blades, except of the upper section near the shroud where some recirculation can be observed (Fig. 5c). Fig. 4 illustrates the corresponding flow field for a much reduced flow rate, equal to 20% of the 'nominal'. The different flow characteristics can be clearly observed: the pressure gradients are lower throughout the domain and the minimum values are higher (Fig. 5a,b). However, a strong recirculation is established within almost the entire blade-to-blade region, which according to the theory rotates in the reverse direction from the impeller. The results for the case of a double volume flow rate (62 m3/h) were finally obtained and are shown in Fig. 5. Now the pressure and the velocity fields exhibit higher gradients but no important recirculation is observed in the impeller (Fig. 6a). However, due to the higher radial flow velocity at the blades inlet, the stagnation line is moved towards the suction side of the blade leading edge, and consequently the minimum pressure is displaced to the opposite pressure side (Fig. 6b).

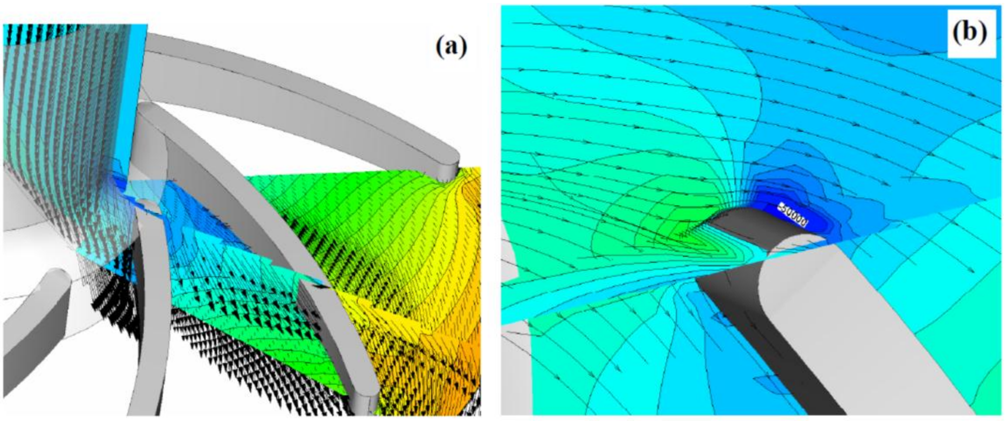


Fig 6 Flow field analysis,Q_u=60m^3/h.

6 IMPELLER PERFORMANCE

The numerical model is then applied to reproduce the performance characteristic curves of the standard impeller at 1500 rpm, covering the entire load range from 20% to 200% of the 'nominal' value. The computed curves are drawn in Fig. 7a and the first observation is that the maximum hydraulic efficiency is achieved for Q ≈ 22,5 m3/h, which is lower than the 'nominal' value. However, a better estimation of the latter for the simulated geometry with the aid of the inlet triangle of velocity vectors gave the value of about 25 m3/h. The net fluid head and the impeller power curves, plotted in Figs 7a, respectively, exhibit a reasonable and smooth pattern, verifying the reliability of the PFC grid technique. As expected, an increase of the volume flow rate above a safety limit deteriorates the suction characteristics of the impeller. The minimum pressure values in Fig. 7b are drastically reduced for Q > 50 m3/h, indicating that intense cavitation may occur in the impeller.

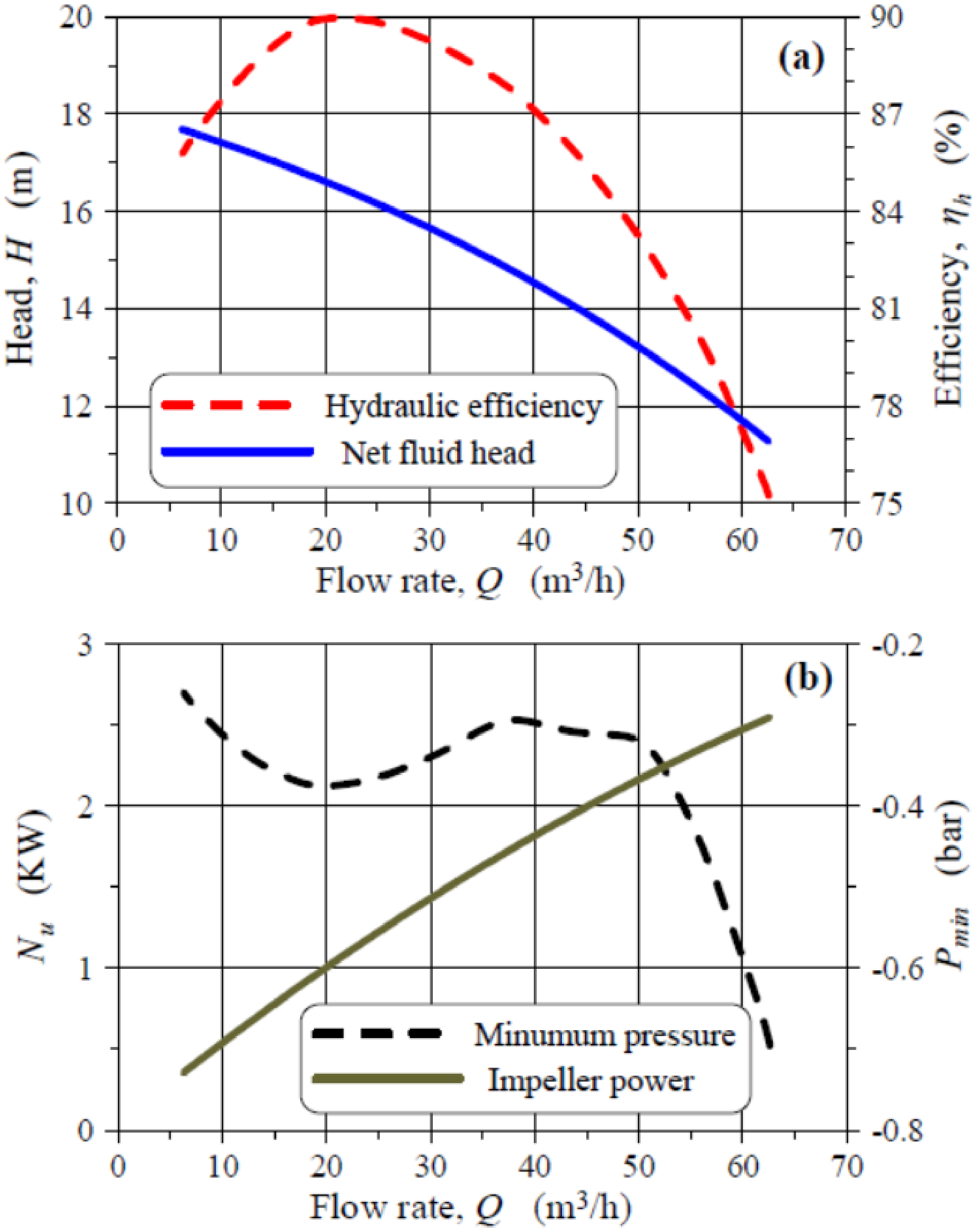


Fig 7a.Flow Rate

7 REFERENCE:

[1] Rakibuzzaman, "A study on multistage centrifugal pump performance characteristics for variable speed drive system", Procedia Engineering 105 (2015) 270 – 275.

[2] S.Shaaban, "Design optimization of a centrifugal compressor vaneless diffuser", S0140-7007(2015)00188-7.

[3] Janusz Skrzypacz, "Numerical modelling of flow phenomena in a pump with a multi-piped impeller", Chemical Engineering and Processing 75 (2014) 58-66,

[4] H.Stel , "Numerical investigation of the flow in a multistage electric submersible pump", Journal of Petroleum Science and Engineering, 136 (2015) 41–54

[5] Wenjie Wang, "Application of different surrogate models on the optimization of centrifugal pump", Journal of Mechanical Science and Technology, 30 (2) (2016) 567-574

[6] Shuai yang, "Dynamic analysis of the pump system based on MOC–CFD coupled method", Journal on Annals of Nuclear Energy, 78 (2015) 60–69

[7] Richard B. Medvitz, "Performance Analysis of Cavitating Flow in Centrifugal Pumps Using Multiphase CFD", Transactions of the ASME, Vol. 124, JUNE 2002

[8] S.Rehman , "Pumped hydro energy storage system of Renewable and Sustainable Energy", Reviews 44 (2015) 586–598.

[9] F.Pugliese, "Characterization of two pumps as turbine for hydro power generation", Renewable Energy 99 (2016) 180-187.

[10] Bikram Singh Solanki , "Centrifugal Pump in Turbine Mode for Small Hydro Power Generation Mechanical Engineering Department", UIT-RGPV, Bhopal, India 25 May 2016, Vol.6, No.3 (June 2016).

[11] Masahiro Nishida, "Effect of Impeller Geometry on Lift-Off Characteristics and Rotational Attitude in a Mono pivot Centrifugal Blood Pump," Artificial Organs 2016, 40(6):E89–E101

[12] M.J.Montazeri, "Multidisciplinary optimization of a pump–fed system in a cryogenic LPE using a systematic approach based on genetic algorithm", Aerospace Science and Technology 49 (2016) 185–196.