Abstract - Mixed flow pumps are a combination of both axial and radial flow pumps. In present days, they are having a wide range of applications. The impeller blade designing of mixed flow pump has always been a difficult task, the reason being it requires a lot of designing parameter to be considered for successful designing. When compared with radial and axial flow turbo-machines, the mixed flow type of turbo-machines always get less importance for these complexities in their designing process. This complexity arises because of the complicated blade and geometry of fluid flow passage. Impeller is considered as one of the most important components in a mixed flow pump for the flow passage. The main objective is to improve the mixed flow pump efficiency and power up to 5-10% by using the method of varying the inlet and outlet blade angle. The three dimensional flow field of the whole flow passage of mixed flow pump in existing impeller and new impeller was optimally designed and numerically simulated using CFD software. The numerically simulation results show that the hydraulic performance of the mixed flow pump. By varying inlet and outlet blade angle with increasing and decreasing 10-20% with respect to design angle, the performance of pump has been analyzed. The modified eight blade design has improved efficiency of 5% when compared with existing six blade design.

Keywords: Mixed Flow Pumps; Impeller Blade; Turbo-Machines; Vane Angle; CFD; Blade Angle

1. INTRODUCTION

A pump is device which is used to displace fluids or sometimes slurries by mechanical work done. Pump is a mechanical positive device generally used for lifting liquids from a lower level to a higher. This will be achieved by creating low pressure at the inlet and high pressure at the outlet of the pump. However, actual work has to be done by the prime movers to enable it to give mechanical energy to the liquid which converts into pressure energy. It is mostly in used in industries and home applications. Centrifugal pumps are the device, which employs centrifugal force to lift the fluid from a lower level to a higher level by developing pressure. The centrifugal pump can move liquid by rotating impellers inside a volute casing. The liquid is passing through the casing inlet to the centre of the impeller where it is picked up by the impeller vanes. In the mixed flow pump, the addition of energy to the liquid occurs when the flow of the liquid takes place in an axial as well as in the radial directions is shown in fig 1. In this type of pump flow of liquid through impeller is a combination of axial and radial directions. The head can be developed partly by the action of centrifugal force and partly by the action of propelling force. These pumps are mostly suitable for irrigation purposes where large amount of water is to be pumped at a lower head.

Fig. 1 Mixed Flow Pump

Mixed-flow pumps play an extremely important role in the economic construction of China. They have been widely used in agricultural irrigation, flood control, urban water supply, the cooling water system of power plant. Over the period of time the design has been carried out according to the various templates. Wislicensus [1] initiated the design of a mixed flow pump impeller. The modification of mixed flow pump impeller was carried out by Myles [2]. Stepanoff [3] gave a design procedure for mixed flow pump impeller. Neumann [4], Gahlot and Nyiri [5] had suggested the one-by-one design procedure for designing mixed flow pumps. Few real 3D inverse design methods had been developed, for example the inverse time marching method [6], the pseudo-stream function method [7], and the Fourier expansion singularity method [8]. These methods are very time-consuming and exhibit some difficulties in correlating the design parameters with the blade geometry (the first two) or convergence problems (the latter). A quasi-3D method had also been proposed [9], which performs a blade-to-blade solution and saves computer time by using only one representative hub-to-shroud surface.

All the above models are based on the in viscid simplified assumption. John [10] had recently developed a fast numerical method for flow analysis and blade design of impellers. The mixed-flow pump studied in this paper was used to take water from a
river by a water company. During the installation and commissioning stage, technicians discovered that the flow rate of this pump was only 2950m³/h when the head is 14m, which was far from meeting the design requirements of the owner (the flow rate shall be 5700m³/h when the head is 14m) [11]. Entrusted by this company, optimal design of this pump was conducted with only replacing the impeller and without replacing the flow passage, fixed guide vane and motor. Impeller is an important flow passage component in a mixed-flow pump. To find out the causes of low hydraulic performance of the original pump, it is necessary to study the flow in the impeller. While the impeller of the pump is operating, the impeller will rotate, and the geometric shape of the flow passage will be distorted. The water flow in the impeller is a complicated three-dimensional turbulent flow. Therefore, observing the flow situations in the impeller through experiment not only will be a waste of time, but also will cause economic losses due to the shutdown. In recent years, with the rapid development of computer technologies and computational fluid dynamics, CFD software has been used to numerically simulate the three-dimensional viscosity flow in the impeller, and optimal design of the impeller on this basis has become a reality.

1.1 Analysis of Mixed Flow Pump

FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within the system. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested. In practice, a finite element analysis usually consists usually consists of three principal steps.

1.2 Generic Steps to Solve any Problems in ANSYS

The following steps are using to solve any problems in ANSYS these are physical model, Boundary conditions and the physical properties.

1.3 Introduction to CFD

Fluid (gas and liquid) flows are governed by partial differential equations which represent conservation laws for the mass, momentum, and energy. Computational Fluid Dynamics (CFD) is the art of replacing such PDE systems by a set of algebraic equations which can be solved using digital computers. Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of Mathematical modeling (partial differential equations), Numerical methods (discretization and solution techniques) Software tools (solvers, pre- and post-processing utilities) CFD enables scientists and engineers to perform numerical experiments (i.e. computer simulations) in a virtual flow laboratory.

2. LITERATURE REVIEW

To simulate rotating machinery in CFD, various approaches for modeling the rotor can be taken. These include approximate models of the rotor such as the pressure jump approach [12] (Shankaran and Dogruoz, 2010; Van der Spuy et al., 2010) and the actuator disk model (ADM) approach. The need for designing a higher efficiency pump with lower energy consumption is increasing with the current advocacy for green power, nowadays. The ADM approach described in-depth by Thiart and von Backström (1993) has been validated by numerous authors (Bredell et al., 2006; Meyer and Kröger, 2001; Van der Spuy et al., 2010). This approach can accurately predict the performance of mixed flow pump for a large range of flow rates. To design and optimize the blade shape, the distribution of the structural parameters, for instance the blade angles, were directly adjusted in previous studies. Unfortunately, this approach is highly dependent on the designers’ experiences. Additionally, a fair number of design parameters would need to be selected to be optimized in order to get a new runner blade that is quite different from the target. The ADM, however, was found to under-predict the performance at lower flow rates further away from the design point (Le Roux, 2010). When the pumps were first used in turbine mode is unclear. In 1931, when Thoma and Kittredge (1931) were trying to evaluate the complete characteristics of pumps, they accidentally found that pumps could be operated very efficiently in the turbine mode. Different pumps suitable in turbine is shown in fig. 2. The turbine mode operation became an important research question for many manufacturers as pumps were prone to abnormal operating conditions. Later in 1941, Knapp (1941) published the complete pump characteristics for a few pump designs based on experimental investigations. In the 1950s and 1960s, the concept of pumped storage power plants, in the range of 50 to 100 MW, was evolved mainly in developed countries to manage the peak power requirements.

In later years, chemical industries became another area for the application of Pump as Turbine (PAT) for energy recovery. Even in water supply networks identical applications of this technology were found. This background gave some momentum to a rich phase of research and then onwards, standard manufactured pumps were studied in turbine mode. In later years, many more techniques were developed by many researchers (Rawal and Kshirsagar, 2007).
3. OBJECTIVES

To improve the mixed flow pump efficiency and power up to 5-10%. Head predicted by CFD analysis is 5 to 10% higher at rated point as well as at part load, than the test result. The nature of Head versus Capacity curve is similar to that of standard mixed flow pump curve. Power predicted by CFD analysis is 5 to 10% higher at rated point. Disc friction power loss calculated using standard method and added to CFX generated power for estimating total power requirement. The nature of Power versus Capacity curve is similar to that of standard mixed flow pump curve.

Efficiency predicted by CFD analysis is higher than the test results. Leakage loss was predicted and volumetric efficiency was determined. Pump efficiency, considering disc-friction loss and leakage-loss was predicted and that is within +5% range, at duty point. At off-design point, variation in efficiency was found.

4. METHODOLOGY

The numerical study was undertaken using the ANSYS-CFX computational fluid dynamics (CFD) software package. A single blade passage was simulated to improve computational efficiency, reducing the time required per simulation. This approach also allowed for validation of the results using the cold flow test rig at Queen’s University Belfast (QUB), which produced an axis-symmetric flow field at the inlet to the turbine.

The test rig was equipped with pre-swirl vanes to create levels of swirl equivalent to what is typically produced by a mixed flow pump turbine housing. Additionally, by adopting this approach any asymmetric flow field effects generated by a volute tongue were not present. The rotor selected for the study was a non-commercial MFT design developed as part of a previous study at QUB and was derived from a state-of-the-art radial flow turbine. The rotor consisted of 9 blades, with $\lambda = 30^\circ$ and an idealized $\beta_b = 30^\circ$.

A typical mixed flow pump turbine is not equipped with a diffuser, however, for this study a diffuser was included to accurately represent the QUB cold flow turbine test rig. The rotor and diffuser were modelled as a single domain, with the diffuser shroud defined as a counter rotating wall to account for its stationary position; this avoided any distorting effect from an interface just downstream of the rotor trailing edges.

The outlet flow properties were extracted from the model at a location one radius downstream of the trailing edge, to negate the effects of the diffuser. The shear stress transport (SST) turbulence model was selected for the simulations as it has been demonstrated to achieve good agreement with experimental validation in previous studies at QUB.

4.1 Materials and Components

The materials used for manufacturing the components of turbine are as follows:

- Impeller – Bronze
- Shaft - Pump steel
- Shaft sleeve - Stainless steel
- Shaft enclosing tube - Pump steel
- Pump base - Cast iron
- Driver pedestal - Rolled steel

In the field of thermoplastics design, there is a growing awareness of the important of stress analysis. In many years, plastics have been used for application in load-bearing structural component in the automotive, aerospace, sporting, and construction industries. Hence, design engineers are increasingly concerned about stress-related problems, typically with the strength, stiffness and life expectancy of their products.

<table>
<thead>
<tr>
<th>S. No.</th>
<th>Content</th>
<th>Steel properties</th>
<th>Bronze properties</th>
<th>Cast iron properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Density</td>
<td>7800 kg/m$^3$</td>
<td>7400-8900 kg/m$^3$</td>
<td>6800-7800 kg/m$^3$</td>
</tr>
<tr>
<td>2</td>
<td>Young’s Modulus</td>
<td>2x10$^5$ N/mm$^2$</td>
<td>1.2x10$^5$ N/mm$^2$</td>
<td>1.7x10$^5$ N/mm$^2$</td>
</tr>
<tr>
<td>3</td>
<td>Poisson’s Ratio</td>
<td>0.333</td>
<td>0.34</td>
<td>0.26</td>
</tr>
<tr>
<td>4</td>
<td>Tensile Strength</td>
<td>460 mm$^2$</td>
<td>500 mm$^2$</td>
<td>200 mm$^2$</td>
</tr>
</tbody>
</table>

4.2 Modeling

Modelling of mixed flow pump is done as with help of solid works software and dimensions are selected from one of the mixed flow pump as shown in fig. 3 and fig. 4. As the pressure is more for the front portion of pump only outer dimensions of mixed flow pump has been considered for modelling. Slots provided in middle of mixed flow pump are used for reducing drag effect in mixed flow pump.

![Fig. 3 3D Design Diagram Of 8 Blades](image)  ![Fig. 4 3D Design Diagram of 6 Blade](image)
5. RESULTS AND DISCUSSION

5.1 Validation of CFD Models

Verify the code to make sure that the numerical solutions are correct. Compare the results with available experimental data (making a provision for measurement errors) to check if the reality is represented accurately enough. Perform sensitivity analysis and a parametric study to assess the inherent uncertainty due to the insufficient understanding of physical processes. Try using different models, geometry, and initial/boundary conditions. Report the findings, document model limitations and parameter settings. The design is imported from the design software and meshing is done in ansys software as shown in fig. 5 and fig. 6.

Fluid flows encountered in everyday life include, Meteorological phenomena (rain, wind, hurricanes, floods, fires), Environmental hazards (air pollution, transport of contaminants), Heating, ventilation and air conditioning of buildings, cars etc and Combustion in automobile engines and other propulsion systems. The goal of verification and validation is to ensure that the CFD code produces reasonable results for a certain range of flow problems.

5.2 Details of Impeller 1

Outlet blade angle 16.280°
Inlet blade angle 21.080°
No. of blades 6

Pressure and velocity diagram of impeller 1 is shown in fig.7 and fig. 8.

5.3 Details of Impeller 2

Outlet blade angle 16.13°
Inlet blade angle 24.40°
No. of blades 6

Pressure and velocity diagram of impeller 2 is shown in fig.9 and fig. 10.
5.5 Model Calculation

Inlet power = \( \frac{2\pi NT}{60 \times 1000} \)

\[
\begin{align*}
\text{Inlet power} &= 2\pi \times 1450 \times 388.122 / 60 \times 1000 \\
&= 58.934 \text{ kW}
\end{align*}
\]

Outlet power = \( (P_0 - P_i) \times Q / 1000 \)

\[
\begin{align*}
\text{Outlet power} &= (215916.8 - 17900.8) \times 0.25 / 1000 \\
&= 49.504 \text{ kW}
\end{align*}
\]

Overall efficiency = \( \frac{\text{Outlet power}}{\text{Input power}} \)

\[
\begin{align*}
\text{Overall efficiency} &= 49.504/58.934 \\
&= 0.84
\end{align*}
\]

Table 2 Calculation of Efficiency

<table>
<thead>
<tr>
<th>Impellers</th>
<th>Inlet power (KW)</th>
<th>Outlet power (KW)</th>
<th>Efficiency %</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller 1</td>
<td>59.93</td>
<td>49.50</td>
<td>84.00</td>
</tr>
<tr>
<td>Impeller 2</td>
<td>57.04</td>
<td>39.93</td>
<td>70.00</td>
</tr>
<tr>
<td>Impeller 3</td>
<td>60.12</td>
<td>52.91</td>
<td>88.00</td>
</tr>
</tbody>
</table>

6. CONCLUSION

From the result we can conclude that, Inlet angle changes will change the range between 16.88° to 24.64° & outlet angle changes will changes range between 36.32° to 54.58°. Maximum efficiency is attained at the point of inlet angle 16.88° and outlet angle 20.26°. So that we can conclude that best design point of impeller of pump that we can give the maximum efficiency of pump that is inlet angle 16.88° and outlet angle 20.26°. Here we can take the only efficiency for the design point of view.

Head predicted by CFD analysis is 5 to 10 % higher than the test result at rated point.

Power predicted by CFD analysis is 5 to 10% higher at rated point.

Efficiency predicted by CFD analysis is higher than the test result. Pump efficiency considering disc friction loss and leakage-loss is predicted and it was found within +5% range, at duty point.

CFD analysis has indicated recirculation zone in the stator at duty point, which has caused reduction in efficiency.

Efficiency is improved by 1% after matching stator angle and changing hub curve profile. Stator blade loading at hub and shroud has improved.

REFERENCES


