CFD ANALYSIS – MULTI STAGE SUBMERSIBLE PUMP

INTRODUCTION

The mixed-flow submersible pumps are used for domestic purpose and also at the commercial level in a large extent, so increasing the efficiency of the pump is an essential part. Design of multi stage pump is really complex due to the fact that the flow inside this pump is inherently complex in nature. With the rapid development of the computer technology, the Computational fluid dynamics (CFD) and the numerical simulation has become an important tool to study flow field inside the pump and predict its performance.

With the aid of the CFD technology, the complex internal flows in water pump impellers can be well predicted and speed up the design procedure. Thus, CFD is an important tool for pump designers since it will reduce the time consuming and expensive experimental procedures.

SCOPE OF THIS RESEARCH WORK

- To predict the performance improvement of Bowl-Impeller Axial Gap in a multi stage mixed flow submersible pump.
- Six different Bowl-Impeller axial gap variations were considered (vary from 2mm, 5mm and 10 mm in Increasing and Decreasing order).
- Taguchi Method is utilized to reduce the number of trials and to optimize the best performance.

REVERSE ENGINEERING PROCESS

The submersible pump is scanned based on the reverse engineering in Coimbatore Industrial Infrastructure Association (CO-INDIA) Avarampalayam, Coimbatore. By using CAD software package solid works V10, the drawing has generated.

CFD ANALYSIS

With a goal of increasing hydraulic efficiency of the pump CFD is utilized in a following systematic way.

VALIDATION STUDY

CFD procedure has lot of variables such as grid count, turbulence model, and discretization scheme, so it is important for a CFD engineer to play between those variables without affecting the solution accuracy. A commercial Computational Fluid Dynamics (CFD) code, ANSYS Fluent V13.0 with a k-ε Realizable turbulence model was used to study the effects of Bowl-Impeller Interaction. Moving Reference Frame (MRF) technique is used to numerically model the rotation of the impeller section to predict the flow behaviour of the pump.

The numerical results are compared with the experimental data of the base model to arrive at the best computational procedure (BCP).

- PERFORMANCE ENHANCEMENT STUDY

The hydraulic performance of the pump highly depends upon the complex configuration of the Bowl-Impeller interaction, number of blades on bowl-impeller, inlet-outlet angles, radial and axial clearance between Bowl-impeller and other various dimensions such as diameter, width etc., In this case axial clearance, number of blades and outlet angle on impeller are taken as the major variables for optimization.

- UTILIZATION OF DOE METHODOLOGY

Taguchi method is a scientifically disciplined mechanism for evaluating and implementing improvements in products, processes, materials, equipment, and facilities. These improvements are aimed at improving the desired characteristics and simultaneously reducing the number of defects by studying the key variables controlling the process and optimizing the procedures or design to yield the best results.

INFLUENCE OF CFD TECHNIQUE IN THIS PROBLEM

This pump highly depends upon the complex configuration of the Bowl-Impeller interaction, and various geometrical parameters. The vertical space between the bowl and impeller centres is called as “Axial gap”. The following results are taken for the existing model with an axial gap of 17mm.

******Using CFD, the results for six different modifications was simulated within less time whereas experiments need lot of time and cost investment to test individual modification itself. No experiment will provide an INSIGHT views like the pressure contour, velocity vectors in different planes and positions as shown in Fig 2, Fig 3, Fig 4 and Fig 5.***

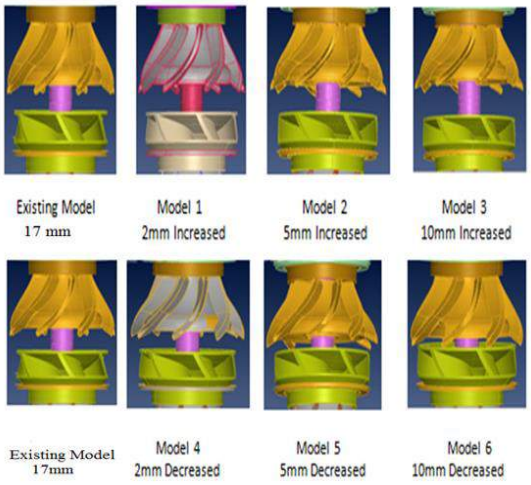


Fig 1 Six different models

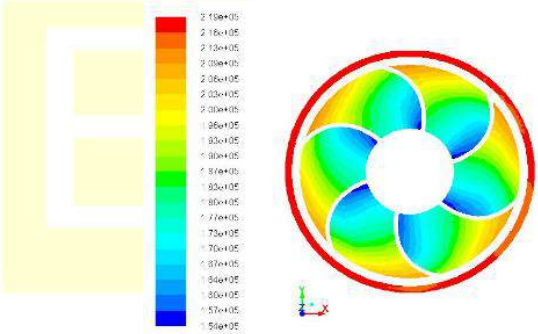


Fig 2 pressure contour-impeller (Pa)

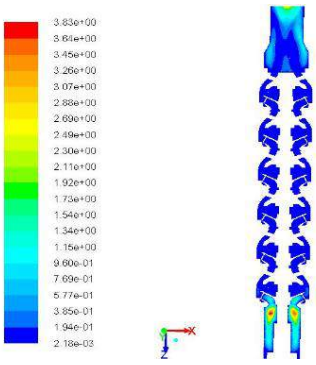


Fig 3 Turbulent viscosity (Kg/m-s)

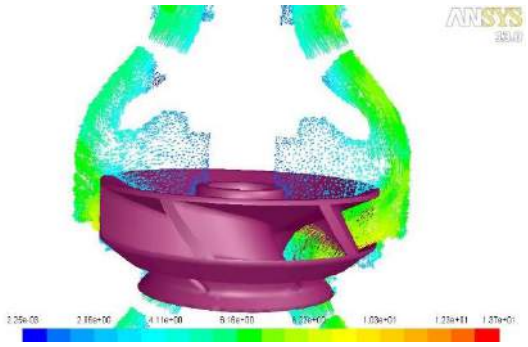


Fig 4 Axial gap of 10 mm

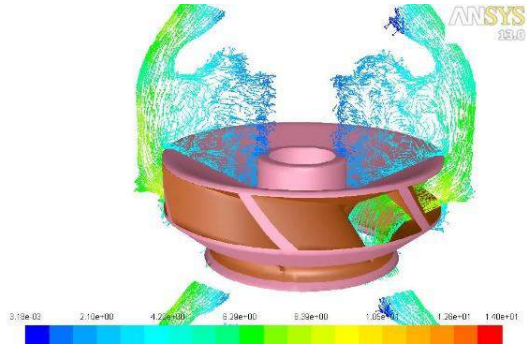
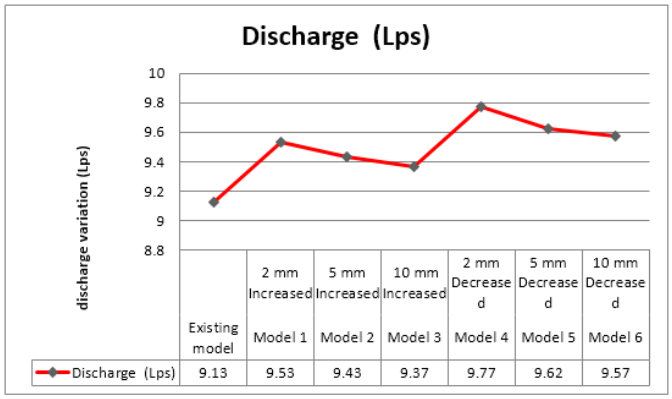
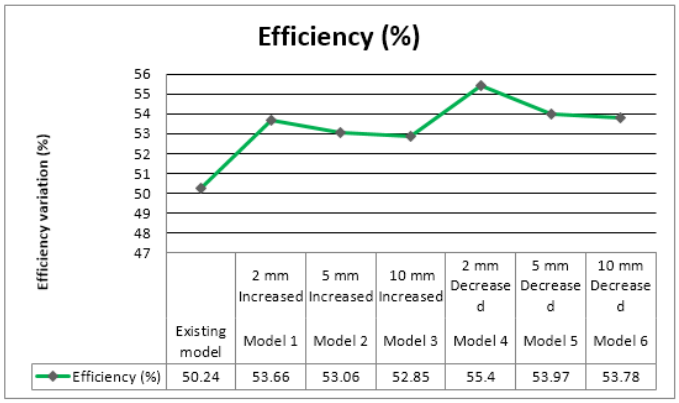


Fig 5 Axial gap of 27 mm

******Above Figures shows the effect of altering the axial gap. Increase in axial gap increases recirculation zone causing a reduced flow rate. And it is impossible to get this much detailed view of velocity profiles when we go for experiments so this is the place where one can see the merits of CFD techniques.***



	Existing model	2 mm Increased Model 1	5 mm Increased Model 2	10 mm Increased Model 3	2 mm Decreased Model 4	5 mm Decreased Model 5	10 mm Decreased Model 6
Discharge (Lps)	9.13	9.53	9.43	9.37	9.77	9.62	9.57



	Existing model	2 mm Increased Model 1	5 mm Increased Model 2	10 mm Increased Model 3	2 mm Decreased Model 4	5 mm Decreased Model 5	10 mm Decreased Model 6
Efficiency (%)	50.24	53.66	53.06	52.85	55.4	53.97	53.78

CONCLUSION

- ✓ A validation study achieving the best practice of CFD and followed for further analysis.
- ✓ An optimization study is tried out using Taguchi method by varying three major parameters such as outlet vane angle in the impeller, number of impeller blades and the axial clearance between bowl and impeller. An optimum configuration is achieved after comparing the results.
- ✓ Among Six different variations an optimum axial clearance is achieved by comparing the simulation results.