A Review Study on CFD Analysis and Experimental study on Impeller of Centrifugal Pump

Alpeshkumar R Patel[1] Neeraj Dubey[2]
[1]PG Student [2]Associate Professor
[1, 2]Department of Mechanical Engineering
[1, 2]Sagar Institute of Research & Technology, Bhopal

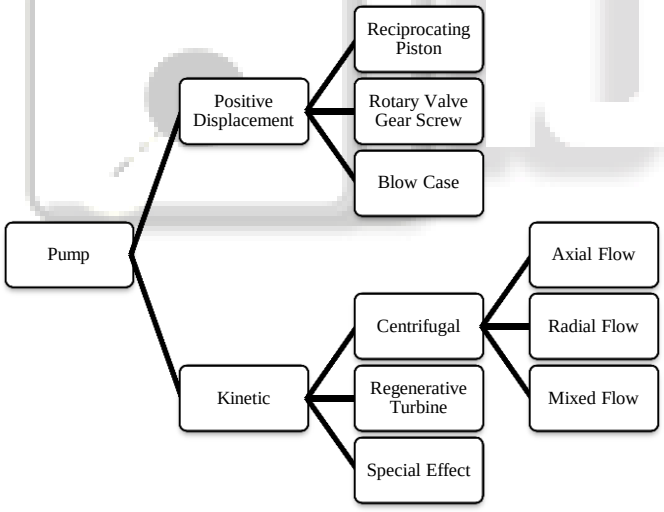
Abstract---A subject of this paper is CFD analysis and experimental study on the impeller of centrifugal pump. The geometrical design data of impeller is measured manually with precision measuring instruments. The impeller's geometry is transferred into 3D CAD modelling software. The paper also presents the result of original geometry and validation of test result and CFD analysis result. The aim of CFD analysis is to optimize the geometry of impeller and improve the head of pump.

I. INTRODUCTION

A pump, a device that expends energy in order to raise, transport, or compress fluids.. A fluids may be liquid or gas. Pumps are classified according to move the fluid.

1) According to force,
2) according to distance and,
3) direct lift

A. *Classification of pumps*

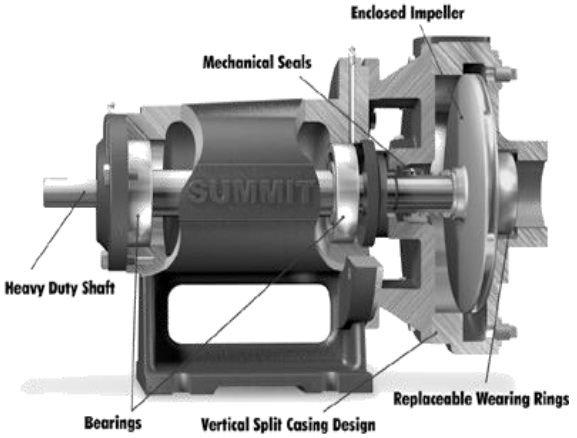


B. *Centrifugal Pump*

A centrifugal pump is a rotodynamic pump that uses a rotating impeller to increase the pressure and flow rate of a fluid. Centrifugal pumps are the most common type of pump used to move liquids through a piping system. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward or axially into a diffuser or volute chamber, from where it exits into the downstream piping system. Centrifugal pumps are typically used for large discharge through smaller heads.

Centrifugal pumps are most often associated with the radial-flow type. However, the term "centrifugal pump" can be used to describe all impeller type rotodynamic pumps including the radial, axial and mixed-flow variations.

C. *Structure of centrifugal pump*



1) *Pump casing*

pump casing include two types: axial split type and radial split type. Most casings of single stage pumps are volute. Radial split casing of multistage pumps are usually ring casings and round casings.

The inner chamber of volute pump casing is generally spiral fluid passage, which is used to collect the liquid thrown out from impeller and lead liquid to the diffusion tube towards the outlet of the pump. Pump casing withstands all working pressures and heat load of the liquids.

2) *Impeller*

impeller is only part to do work. Pump does work on liquid by impellers. Impeller can be classified in three types: closed type, open type, and semi-open type. The closed impeller is made up of blade, the front cover plate and rear cover plate. The semi-open impeller is made up of blade and rear cover plate. The open impeller is only made up of vane. The efficiency of closed impeller is higher than open impeller.

3) *Seal ring*

The function of seal ring against inner leakage and external leakage. The seal ring is made of wear resistance material is mounted on front and rear cover plate and pump casing. The seal ring can be replaced after wearing down.

Shaft and bearing: one end of shaft is used for fixing impeller; the other end is used for installing coupling. Based on pump size, bearing can adopt rolling bearing and plane bearing.

4) *Shaft seal*

shaft seal generally include mechanical seal and packing seal. The pump is generally designed that can be installed with not only the packing seal, but also mechanical seal.

D. Working principle of centrifugal pump

Centrifugal pump mainly is composed of impeller, shaft, casing, shaft seal, etc. The pump casing needs to be filled with liquids before starting. When pump shaft and impeller rotate, driven by prime motor, liquids move in a circle and at same time thrown out from impeller centre under the action of centrifugal force. So the liquids obtain pressure energy and velocity energy from impeller. Partial velocity energy converted in to static pressure energy. When liquids flow towards liquid outlet via volute. After liquids are thrown out from impeller, impeller centre shall be low pressure area. Which has pressure difference with the pressure of the inhaled liquid surface? Therefore, liquids are continuously inhaled and discharge with a certain pressure.

E. What is CFD

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. In order to shorten the design periods and lowering the manufacturing, prototyping and test costs of the pump, commercial Computational Fluid Dynamics (CFD) software is used in the design procedure. The main aim of this study by means of applying numerical experimentation to the designed pump is CFD code integration into the design procedure and verification of the design before the pump is produced. The CFD code is used to obtain pump characteristics curves such as head vs. flow rate and efficiency vs. flow rate.

Working in CFD is done by writing down the CFD codes. CFD codes are structured around the numerical algorithms that can be tackle fluid problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces input problem parameters and to examine the results. Hence all codes contain three main elements:

1) Pre-processing.
2) Solver
3) Post – processing

II. Literature Review

Michal Varchola (2012) et al represented work on a numerical solution of a mixed - flow pump geometry with respect to a distribution of a static pressure. The hydraulic projection of an impeller is very sensitive in terms of the overall efficiency as well as the position of the best efficiency point. The blade angle affects several hydrodynamic parameters of the pump especially the position of flow rate and the shape of pump characteristics. The peak efficiency is achieved by vary the inlet and outlet angle.

Peter Hlbocan (2012) et al represented work on the Prime Geometry Solution of a Centrifugal Impeller Within a 3D Setting. It concludes that the optimum result obtained by changing the meridional cut of impeller which increased the mass flow rate and efficiency of the pump.

Maitelli (2010) et al represented work on simulation of flow in a centrifugal pump of EPS system using computational fluid dynamics. It presents a 3D simulation of the stationary flow in the impeller and stator of a mixed centrifugal pump using Computational Fluid Dynamics (CFD) techniques and a commercial software, ANSYS® CFX® Release 11.0. Three conditions were simulated to obtain the pressure fields in the impeller and stator in a stage of the pump. The first condition was the impeller simulation with the blades length equivalent to the real model. The second option was tested for the condition of complete connection of pump, impeller and Diffuser, with the real blades length. In the Third option the impeller and the diffuser had Their external radii increased by four (4) mm.

A. Manivannan (2010) et al represented work on Computational fluid dynamics analysis of a mixed flow pump impeller. Based on the detailed design and CFD analysis of the mixed flow impeller, the following conclusions are derived.

- The mixed flow pump the best efficiency point of the pump is found to be 11 lps.
- The existing impeller, the head, power rating and efficiency ar e found out to be 19.24 m, 9.46 kW and 55% respectively.
- The impeller 1, the percentage increase in the head, power rating and efficiency are 3.22%, 3.9% and 7.27% respectively.
- The impeller 2, the percentage increase in the head, power rating and efficiency are 10.29%, 7.61% and 10.91% respectively.
- The impeller 3, the percentage increase in the head, power rating and efficiency are 13.66%, 12.16% and 18.18% respectively.

Based on the above it is concluded that impeller 3 gives better performance. Thus CFD analysis is an effective tool to calculate quickly and inexpensively the effect of design and operating parameter of pump. By properly designing pump impeller the efficiency of pump can be improved.

From the study of literature review it is concluded that few work was done on centrifugal pump impeller. Some researcher's works on geometrical parameter like inlet and outlet angle, meridional cut of impeller. Now in this present work the thickness of blade has been varied with different flow rate and the characteristic curve of pump obtained.

III. Conclusions

Computational fluid dynamics method is very economical, easier and less time consumption compare than conventional method.

References

[1] A.Manivannan, "Computational fluid dynamics analysis of a mixed flow pump impeller", International Journal of Engineering, Science and Technology, www.ijestng.com(2010).
[2] Maitelli, C.W.S.de p, Bezerra, V.M.deF, da Mata, w. "Simulation of flow in centrifugal pump of ESP systems using computational fluid dynamics", Brazilian Journal of Petroleum and Gas, www.portalabpg.org(2010).
[3] Perez J. , Chiva S., Segala W., Morales R., Negrao C., Julia E., Hernandez L. "Performance analysis of flow in a impeller-diffuser centrifugal pumps using CFD: Simulation and experimental data comparisons". V

European Conference on Computational Fluid Dynamics. (2010).
[4] Peter Hlbocan, Michal Varchola 'Prime Geometry Solution of a Centrifugal Impeller within a 3D setting" www.sciencedirect.com(2012).
[5] Michal Varchola,Peter Hlbocan, "Geometry Design of a Mixed Flow Pump Using Experimental Result of on Internal Impeller Flow"(2012).

