International Journal of Recent Trends in Engineering & Research
ISSN(ONLINE) : 2455-1457
IMPACT FACTOR : 3.344

PERFORMANCE ANALYSIS OF MIXED FLOW PUMP IMPELLER USING CFD

T.Sivakumar[1], P.Aravind[2], U.Aravindh Kumar[3], R.Balamurugan[4], V.Bharathi Priyan[5]
[1]*Associate Professor, Mechanical Engineering, Kathir College of Engineering*
[2,3,4,5]*Students, Mechanical Engineering, Kathir College of Engineering*

Abstract —Centrifugal pumps are probably among the most often used machinery in industrial facilities as well as in common life. After being invented they passed long evolutionary development until they became accessible for various applications. Centrifugal pump design is well facilitated by the use of Computational Fluid Dynamics (CFD). But still some problems are arising to improve the pump performance. This improvement can be achieved by making geometrical changes in design of an impeller. In this study, different techniques for improving centrifugal pump performance by changing impeller geometry are discussed.
Keywords—Pump Performace, Blade angles, Motor, CFD

I. INTRODUCTION

Centrifugal pump is widely used in rural area, for pumping underground water, field irrigation as well as geothermal utilization. The pump performance parameters are expressed in the form of various characteristics such as head, discharge and power consumption, and efficiency are important for pump design. A centrifugal pump converts mechanical energy from a motor to energy of a moving fluid. A portion of the energy goes into kinetic energy of the fluid motion, and some into potential energy, represented by fluid pressure (hydraulic head) or by lifting the fluid, against gravity, to a higher altitude. Increase in performance will lead to increase the head and discharge of the pump, but this improvement affects the power consumption. As per costumers demand of pump performance for limited power consumption is difficult to achieve.

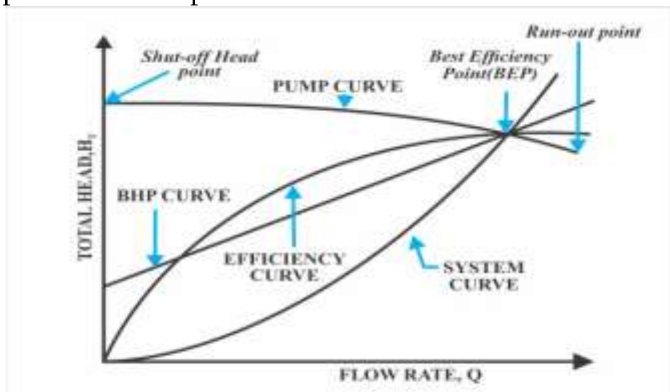

Figure 1.1. Pump Performance Curve

II. COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern this process using a numerical process.

2.1. History of CFD

Computers have been used to solve fluid flow problems for many years. Numerous programs have been written to solve either specific problems, or specific classes of problems. From the mid-1970's, the complex mathematics required to generalize the algorithms began to be understood, and general purpose CFD solvers were developed. These began to appear in the early 1980's and required what were then very powerful computers, as well as an in-depth knowledge of fluid dynamics, and large amounts of time to set up simulations. Consequently, CFD was a tool used almost exclusively in research.

2.2. Steps followed in CFD

- Fluid domain extraction
- Surface meshing
- Volume mesh
- Solving the CFD problem
- Post processing
- Report generating

2.3. Building a mesh

Mesh generation is the practice of generating a polygonal or polyhedral mesh that approximates a geometric domain. The term "grid generation" is often used interchangeably. Typical uses are for rendering to a computer screen or for physical simulation such as finite element analysis or computational fluid dynamics. The input model form can vary greatly but common sources are CAD, NURBS, B-rep, STL (file format). The field is highly interdisciplinary, with contributions found in mathematics, computer science, and engineering.

Three-dimensional meshes created for finite element analysis need to consist of tetrahedral, pyramids, prisms or hexahedra. Those used for the finite volume method can consist of arbitrary polyhedral. Those used for finite difference methods usually need to consist of piecewise structured arrays of hexahedra known as multi-block structured meshes. A mesh is otherwise a discretization of a domain existing in one, two or three dimensions.

2.4. Surface meshing

This package provides a function template to compute a triangular mesh approximating a surface. The meshing algorithm requires to know the surface to be meshed only through an oracle able to tell whether a given segment, line or ray intersects the surface or not and to compute an intersection point if any. This feature makes the package generic enough to be applied in a wide variety of situations. For instance, it can be used to mesh implicit surfaces described as the zero level set of some function. It may also be used in the field of medical imaging to mesh surfaces described as a gray level set in a three dimensional image.

2.5. Volumetric meshes

Volumetric meshes are a polygonal representation of the interior volume of an object. Unlike polygon meshes, which represent only the surface as polygons, volumetric meshes also discretize the interior structure of the object. One application of volumetric meshes is in finite element analysis, which may use regular or irregular volumetric meshes to compute internal stresses and forces in an object throughout the entire volume of the object.

2.6. Post processing

The post-processor is the component used to analyze, visualize and present the results interactively. Post-processing includes anything from obtaining point values to complex animated sequences.

III. METHODOLOGY AND RESULT

3.1. Boundary conditions:

A computational fluid dynamics problem is defined under the limits of initial and boundary conditions. For implementation of boundary conditions when we construct a staggered grid we add an extra node across the physical boundary in order to get,

- The nodes just outside the inlet of the system are used to assign the inlet conditions.
- The physical boundaries can coincide with the scalar control volume boundaries.

Most common boundary conditions used in computational fluid dynamics are

- Intake conditions
- Symmetry conditions
- Physical boundary conditions
- Cyclic conditions
- pressure conditions
- exit conditions

3.2. Procedure of CFD analysis

- 3D Model of Impeller is generated in SOLIDWORKS 2009 as per above given Drawing.
- Our CFD Analysis method is Cavity Patten so we have to create Cavity model of below impeller.
- Save above Cavity model in *. IGES Format for Importing into ANSYS Workbench Mesh Module for Meshing.
- Import above Cavity model in ANSYS Workbench Mesh Module.
- Meshing of Impeller:-

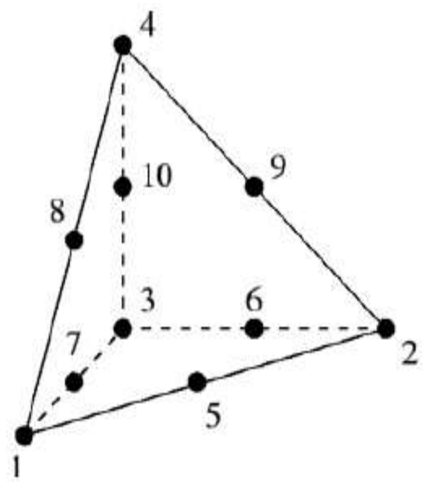

Figure 3.1. Tetrahedral Element

3.2.1. Existing Impeller

Table 3.1. Existing Impeller data

Sr. no.	Parameters	Data
1	Number of blade	8
2	Blade inlet angle	40˚
3	Blade outlet angle	29˚

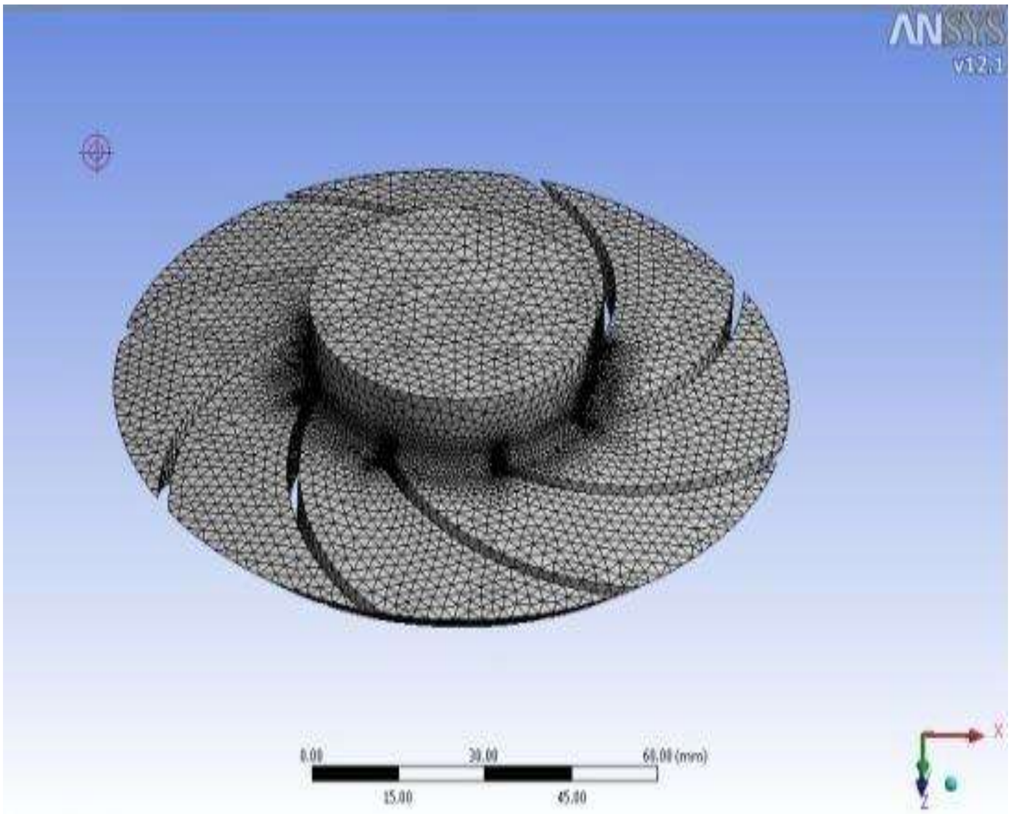

Figure 3.2. Meshing impeller cavity

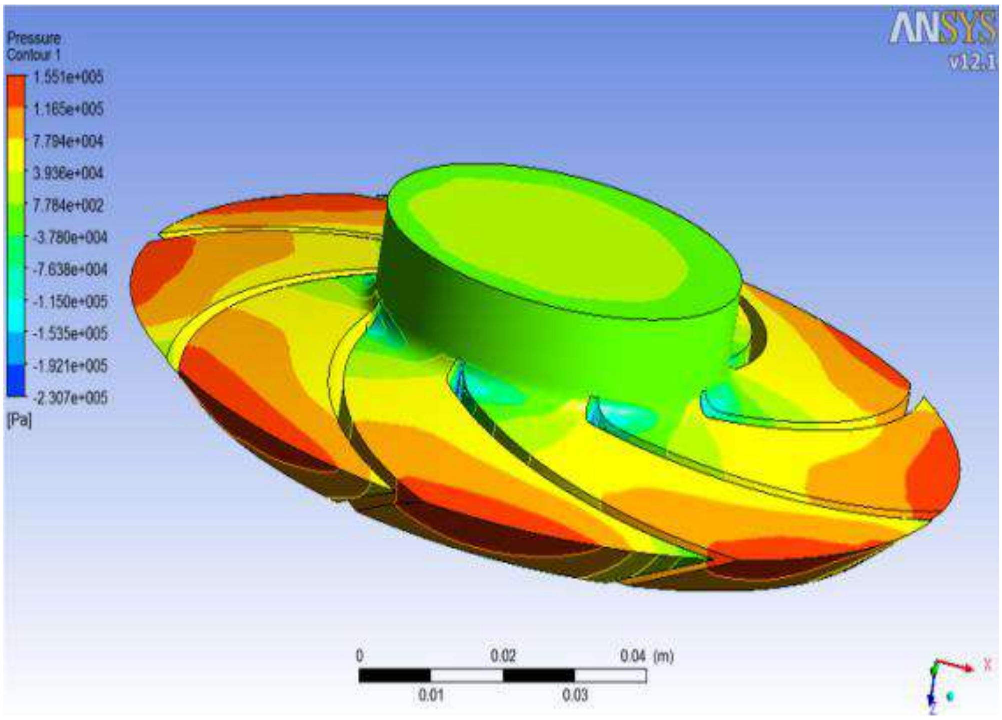

Figure 3.3. Full domain pressure contour impeller 1

3.2.2. Existing impeller data

- Number of Blades = 8
- Outlet pressure = 0.683 x 10^5Pa
- Inlet Pressure = -8.04 x 10^4Pa
- Head = 15.46 m

- Torque = 11.91 Nm
- Velocity = 20.04 m/s
- Power consumed = 3490.54watt
- Efficiency = 79.83%

3.2.3. Modified impeller

Table 3.2. Modified Impeller Data

S no.	Parameters	New data
1	Number of blade	6
2	Blade inlet angle	38˚
3	Blade outlet angle	27˚

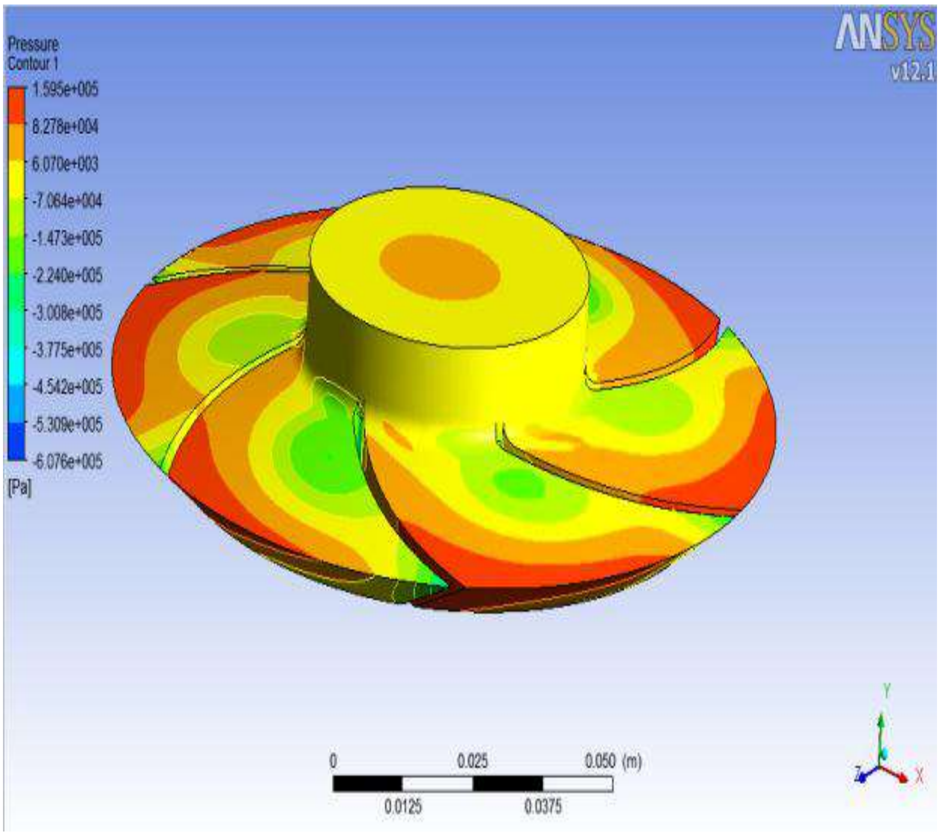

Figure 3.4. Full domain pressure contour impeller 2

3.2.4. Modified impeller data:

- Number of Blades = 6
- Outlet pressure = 1.021 x 10^5Pa
- Inlet Pressure = -9.982 x 10^4Pa
- Head = 20.58 m
- Torque = 9.01 Nm
- Velocity = 24.35 m/s
- Power consumed = 2546.62watt
- Efficiency = 83.34%

IV. CONCLUSION

- Impeller trimming will lead to affect the strength of the impeller.
- Changing impeller blade inlet and outlet angles gives better performance but blade angle should have in suitable value. Larger value creates vacuum or smaller value increases the clogging of water inside the impeller.
- The 9 number of analysis are done for those sets of parameters. Analysis values of performance and the set of parameter 42⁰ inlet angle, 29⁰ outlet angle and 6 number of blade predicated. The Analysis software predicated head is 19.65m, power consumed is 2524.49w, and efficiency is 79.83%
- The suggested set of parameter validation analysis has been done and compared with predicated value. The value of Validation analysis for mixed flow pump head is 20.58m, power consumed is 2546.62w and efficiency is 83.34%.
- These analysis values of mixed flow pump head, power consumed and efficiency are very closer to the predicated by ANSYS software16 values.
- The mixed flow pump impeller is increases the number of blade which gives the maximum head, efficiency and minimum power consumed. Also outlet angle is same, inlet angle is increases so, that maximum head and efficiency.

REFERENCE

1. Kiran patel and N. Ramakrishnan, "computational fluid dynamics Analysis of mixed flow pump"
2. A.Manivannan, "computational fluid dynamics Analysis of mixed flow pump impeller" International Journal of Engineering, Science and Technology, Vol2, No.6,2010,pp.200-206
3. Mandar tabib, Graeme lane, William yang and m Philip schwarz "cfd simulation of a solvent extraction pump mixer unit: evaluating large eddy simulation and rans based models", Seventh International Conference on CFD in the Minerals and Process Industries,CSIRO, Melbourne, Australia9-11 December 2009.
4. Vasilios a. grapsas, john s. anagnostopoulos and dimitrios e. papantonis, "experimental and numerical study of a radial flow pump impeller with 2d-curved blades" proceedings of International Journal of Emerging Trends in Engineering and Development Issue the 5th iasme / wseas international conference on fluid mechanics and aerodynamics, athens, greece, august 25-27, 2007.
5. Maitellic. w. s. de p.1; bbezerra, v. m. de f.; cda mata,w. "simulation of flow in a centrifugal pump of esp systems using computational fluid dynamics" brazilian journal of petroleum and gas | v. 4 n. 1 | p. 001-009 | 2010 | issn 1982-0593.
6. Michal varcholaa, peter hlbocanb, b*, "geometry design of a mixed flow pump using experimental results of on internal impeller flow" international scientific and engineering conference "hervicon-2011 procedia engineering 39 (2012) 168 – 174.
7. Pramesh kumar1, h.l.tiwari2, dr. v. prashad3, dr. v.k.gahlot4, "design of francis type mixed flow pump impeller using C++" journal of engineering research and studies e-issn 0976-7916.
8. J.manikandan, "performance evaluation of mixed flow pump using computational fluid dynamics" european journal of scientific research issn 1450-216x vol.80 no.4 (2012), pp.479-486
9. Milan sedlar, "numerical investigation of flow in mixed-flow pump with volute" jana sigmunda 79, 78350 lutin, czech republic

Books

10. V. K. Jain
    i. Vertical turbine, mixed flow and propeller pumps
11. By G. K. Sahu
    i. Principles of axial and mixed flow pumps