

Research Article

Numerical Flow Simulation of Centrifugal Pump in ANSYS and OpenFOAM

Mohammad Abdul Habib^{Å*}, Shivprakash B. Barve^Å, S. P. Shisode^Å and Savita U Shinde^Å^ÅDepartment of Mechanical Engineering, MAEER's MIT College of Engineering, Pune, IndiaAccepted 10 March 2014, Available online 01 April 2014, **Special Issue-3, (April 2014)**

Abstract

A centrifugal pump is a rotodynamic pump that uses a rotating impeller to increase the pressure of a fluid. The fluid enters the pump impeller along or near to the rotating axis and is accelerated by the impeller, flowing radially outward into a diffuser or volute chamber (casing), from where it exits into the downstream piping. Computational fluid dynamics deals with fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. However, since experiments are limited by the facilities and the costs, computational fluid dynamics is also used to complement the experiments. The lead times and costs of new designs may then be substantially reduced. To use the computational fluid dynamics the various tools are available among them few are commercial and few are open source. For commercial tools one has to pay a lot while for open source tool it is free ware. In this project work the analysis of centrifugal pump in commercial tool i.e. ANSYS and in open source i.e. Open FOAM is carried out. Numerical solutions of the rotor-stator interaction of Centrifugal Pump using ANSYS CFX and OpenFOAM-1.5-dev were investigated.

Keywords: ANSYACFX, OPENFOAM.

1. Introduction

Pumps are designed to increase the pressure of a fluid. This principle is used in hydraulic pumps, ventilating fans and blowers since the earliest ages. According to Reti, the Brazilian soldier and historian of science, the first machine that could be regarded as a centrifugal pump was a mud lifting machine in 1475 in a treatise by the Italian Renaissance engineer Francesco di Giorigio Martini. Real centrifugal pumps did not appear until the late 1600's, when Denis Papin made one with straight vanes. The curved vane was invented by the British inventor John Appold in 1851. Afterward, due to the increasingly larger number of engines required for vehicle and aircraft propulsion, the centrifugal pumps have developed greatly. Due to the centrifugal pumps could be designed smaller for the same efficiency than the other pumps; they were preferred as the main element of engines. In principle, the centrifugal pumps use a rotating impeller with blades to give rotation to the fluid, which is sucked through an inlet pipe. To optimize the design of centrifugal pumps, a lot of measurements are carried out. However, since experiments are limited by the facilities and the costs, computational fluid dynamics (CFD) is also used to complement the experiments. The lead times and costs of new designs may then be substantially reduced. This project work uses the commercial and open-source computational fluid

dynamics software i.e. ANSYS and OpenFOAM respectively to analyze the centrifugal pumps in steady state conditions. The report covers the relevant numerical methodologies together with the associated numerical approach of computational models and describes the simulation set-up models in ANSYS and OpenFOAM. In the end, the obtained dynamic flow results of several deferent numerical solutions are presented together with the comparison against the experimental performance.

1.1 Centrifugal pump parameters

The various parameters that are required to understand the centrifugal pump in details is mention in following section.

- 1) Suction head (HS): The suction head is the available head at the suction nozzle of the pump.
- 2) Discharge head (HD): The discharge head is the vertical distance from the centreline of the pump (this would be the shaft on a horizontal pump) to the level in the discharge vessel.
- 3) The total head (HT): The total head is work of the pump. The total head is measure of the pump's ability to raise the liquid to a given height. The total head is measure of the pump's ability to develop a given discharge pressure. The total head is discharge elevation minus the suction elevation. The discharge head minus the suction head. The discharge head plus

*Corresponding author: Mohammad Abdul Habib

- the suction lift. The total head is discharge absolute pressure reading minus the suction absolute pressure reading.
- 4) Pump Power Output (Po): The power transferred to the liquid at its passage through the pump.
 - 5) Pump Power Input (PI): The power measured at the pump coupling. Pump input or brake horsepower (BHP) is the actual horsepower delivered to the pump shaft.
 - 6) Efficiency (η): Efficiency of pump can be defined as the ratio of the pump energy output to the energy input applied to the pump shaft.
 - 7) Specific Speed (n): The specific speed of centrifugal pump is define as the speed of a geometrically similar pump which would deliver one cubic meter of liquid per second against a head of one meter.

Net Positive Suction Head (NPSH): Gauge reading in meters taken on the suction nozzle referred to the pump centre line, minus the vapour pressure in meters corresponding to the temperature of the liquid plus the velocity head at the point. NPSH as well as inlet total head, is referred to the reference plane. It is necessary to make a distinction between: The NPSH required at given flow and speed of rotation for a given pump. This is a function of pump desires and varies from one design to another. For a cavitations free performance of a pump, it is necessary that available NPSH be higher than required NPSH. The available NPSH represents the difference between the existing absolute suction head and the vapour pressure at the prevailing temperature

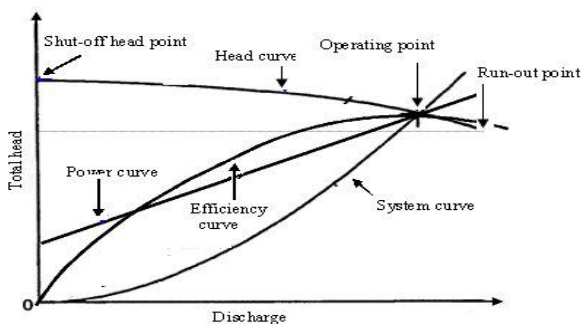


Fig.1 Performance curves of centrifugal pump

1.2 Objective of the project work

The main objective of the project is to analyze the centrifugal pump in ANSYS CFX and OpenFOAM and validate results obtained by both with the experimental results. ANSYS CFX is commercial software while OpenFOAM is open-source software; doing the analysis in both software of centrifugal pump will make a big difference in cost reduction if the OpenFOAM predicting the promising results with the experimental analysis. The various aspects of ANSYS and OpenFOAM are studied in detail. The points that have focused on ANSYS CFX and OpenFOAM are as follow:

- i. Pre processing

- a. Meshing
- b. How to set up a CFD calculation?
- c. Import possibilities from other programs (ICEM, CFX, Fluent)
- ii. Solver
 - a. Convergence
 - b. Monitoring
 - c. Mesh requirements
- iii. Post processing
 - a. Possibilities of post processing
 - b. Export to other program

2. Numerical Approach in ANSYS CFX

In this chapter, the some basic idea of computational fluid dynamics technique for turbo-machinery cases is discussed. The computational meshing of centrifugal pump in ANSYS ICEM and its various parameters to considered while meshing is discussed. Lastly the case set up of centrifugal pump in ANSYS CFX is given. Computational fluid dynamics techniques for turbo machinery cases: This literature summarizes the most important knowledge and experience a CFD engineer needs in order to perform CFD simulations of turbo machinery components.

2.1 2D, quasi-3D or 3D

2D simulations are often used in the early design phase in order to obtain a typical 2D section of a blade. For cases with many long blades or vanes, like low-pressure turbines, a 2D simulation can also provide reasonable results. If the area of the flow-path changes significantly in the axial direction it might be necessary to instead make a quasi-3D simulation. A quasi-3D simulation is a 2D simulation in which extra source terms are used to account for the acceleration or deceleration caused by a changing channel height or growing end-wall boundary layers. Codes focused on turbo machinery applications often have the possibility to perform quasi-3D simulations, but most general purpose CFD codes cannot do this type of simulations, or require user coding to implement the correct source terms in the equations.

2.2 Transient or stationary

Most turbo machinery simulations are performed as stationary simulations. Transient simulations are done when some kind of transient flow behaviour has a strong influence on the global flow field. Examples of transient simulations are detailed simulations of rotor stator interaction effects, simulations of large unsteady separations etc. Sometimes when to perform a steady stationary simulation it can see tendencies of unsteady behaviour like for example periodic vortex shedding behind blunt trailing edges. This is often first seen as periodical variations of the residuals. If the unsteady tendencies are judged to not affect the overall simulation

results it might be necessary to coarsen the mesh close to the vortex shedding or run a different turbulence model in order to make the simulation converge.

2.3 Boundary condition

Describe different types of boundary conditions and when they should be used:

- Total pressure in, static pressure out
- Absorbing boundary conditions
- Mass flow inlet, static pressure outlet.

To put this BC it must allocate inlet surface enough far away from the turbo machine impeller. The velocity distribution is not constant over the entire surface.

2.4 Turbulence inlet conditions

For two-equation turbulence models two different turbulence values need to be specified on the inlet. The most common way of specifying the inlet turbulence variables is to give an inlet turbulence level and length-scale. For simulations where the incoming boundary layers are estimated to not have any significant effect on the secondary flows and global flow-field downstream it is usually sufficient to specify a constant incoming turbulence level and length-scale.

2.5 Turbulence modeling

For attached flows close to the design point a simple algebraic model like the Baldwin- Lomax model can be used. Another common choice for design-iteration type of simulations is the one-equation model by Spalart-Allmaras. The big advantage with both the Baldwin-Lomax model and the Spalart-Allmaras model over more advanced models is that they are very robust to use and rarely produce completely unphysical results. In Numerical approach in ANSYS CFX 16 order to accurately predict more difficult cases, like separating flows, rotating flows, flows strongly affected by secondary flows etc.

2.6 Near-wall treatment

For on-design simulations without any large separated regions it is often sufficient to use a wall-function model close to the wall, preferably with some form of non-equilibrium wall-function that is sensitized to stream wise pressure gradients.

2.7 Numerical considerations

At least a second order accurate scheme for the flow variables can be used. Some codes require a first order scheme for the turbulent variables in order to converge well. It might be sufficient with a first order scheme only

on the turbulence variables, but a second order scheme is of course preferable.

3. Computational meshing of centrifugal pump

The geometry is created by shell scripting and meshing is done in ANSYS ICEM CFD. ANSYS ICEM CFD provides advanced geometry acquisition, mesh generation, mesh optimization, and post-processing tools to meet the requirement for integrated mesh generation and post processing tools for today’s sophisticated analyses. Maintaining a close relationship with the geometry during mesh generation and post-processing, ANSYS ICEM CFD is used especially in engineering applications such as computational fluid dynamics and structural analysis

3.1 Using and placing the equations

Please not that all the equations must be written in MS word not in Mathtype as they are pasted as images which are not clear when the paper is converted in pdf. An example is given as under.

$$RPi = \frac{\sum_k(X_{ik}-CL_k)}{S_k} \tag{1}$$

Where X_{ik} is measured valued of property -k- using inoculants -i-;
 CL_k is average value for property set -k-;
 S_k is standard deviation from the set.

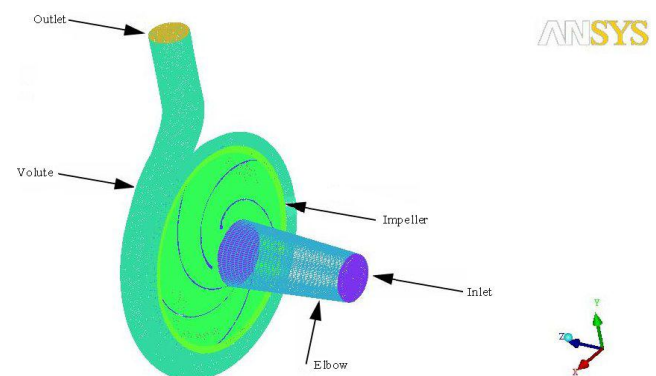


Fig.2.Mesh of centrifugal pump

3.2 Computational mesh of Elbow

The Elbow part of centrifugal pump is mesh with Multi-block structured mesh shown in Mesh statistics of Elbow

- Number of Nodes: 72000
- Number of Elements: 66816
- Hexahedra: 66816

3.3 Computational mesh of Impeller

The impeller of centrifugal pump is mesh with Multi-block structured mesh. To capture boundary layer the fine mesh is done at the blade surfaces i.e. leading & trailing edge. The different views of Multi-block structure mesh are shown:

Mesh statistics of Impeller

- Number of Nodes: 1617415
- Number of Elements: 1533420
- Hexahedra: 1533420

3.4 Computational mesh of volute

Unstructured meshes are used for more complex and odd geometries where a structured mesh is difficult to create. Typical examples where unstructured meshes are often used are blade tip regions, areas involving leakage flows and secondary air systems, film cooling ducts etc. The volute of centrifugal pump is of complex geometry hence mesh with unstructured mesh. To capture boundary layer, the fine mesh is done at the inside of

volute by providing the prism layer and at the tongue section of volute the density of mesh is increase to check whether stagnation point is matching at tongue or not.

- Number of Nodes: 329879
- Number of Elements: 967092
- Tetrahedral: 513627
- Wedges: 453465

4. Discretisation in ANSYS CFX

The term discretisation means approximation of a problem into discrete quantities. The finite volume method and others, such as the finite element and finite difference methods, all are adopting numerical discretisation. Analytical solutions to the Navier-Stokes equations exist for only the simplest of flows under ideal conditions. To obtain solutions for real flows, a numerical approach must be adopted whereby the equations are replaced by algebraic approximations that can be solved using a numerical method.

Case Set-up of centrifugal pump

As the three parts created are in ANSYS ICEM i.e. Elbow, Impeller and Volute similarly three domains are created in ANSYS CFX Pre and it view is shown. The set up for all three domains is different and is explain below. The analysis of pump is steady state. Initially the ANSYS CFX Pre is opened and then meshes are open one by one by using the options keep original mesh or by merging the mesh. Hence for this the geometry is created with the absolute values from the origin.

5. Results and Discussion

In this section, the numerical results obtained by the ANSYS CFX and OpenFOAM are discussed. Various contour plots are also observed in both the cases and their comparisons is also done. The results obtained from the experimental analysis are compared with the ANSYS CFX and OpenFOAM too.

5.1 Residuals in ANSYS CFX and OpenFOAM

A measure of how well the solution is converged can be obtained by plotting the residuals for each equation at the end of each time step. A reasonably converged solution requires a maximum residual level no higher than $5.0E-4$. The RMS residual is obtained by taking all of the residuals throughout the domain, squaring them, taking the mean, and then taking the square root of the mean. The Maximum Residuals can be displayed in the convergence history plots by selecting a specific monitor in Monitor Settings.

5.2 $y+$ value

$y+$ indicates whether the boundary layer is capture or not. Its value depends upon the mesh fineness where the boundary layer is to capture. In this case the boundary layer is to capture at the blades and volute wall. The $y+$ values at the volute walls and blades are less than 50 hence it shows the boundary layer is captures which are shown. The mesh for the ANSYS CFX and OpenFOAM is same the $y+$ values are expected to be same.

5.3 Velocity variations

Velocity vector plots are observed to get the clear picture of flow inside the centrifugal pump. The velocity vector plot also reflects whether the separation of flow and recirculation occurs or not. The velocity vector plot at 0.15, 0.5 and 0.85 span of impeller is observed to get the clear vision of the flow along turbo surface width of the impeller. Looking at the velocity vector plot, the velocity vector plot is nearly same but there is small recirculation in OpenFOAM plots. Near the trailing edge there is more variation in the values due to interaction between the casing walls and impeller, but near the leading edge the variation is less.

5.4 Pressure variations

Pressure plots are observed to see the variation in the pressure throughout the domain. As the suction is created at the inlet so the pressure at the inlet should be less than any other sections. The pressure increases as the fluid is flow away from the impeller i.e. the pressure has to increase in volute. The pressure contour plots at 0.15, 0.5 and 0.85 span of impeller is observed to get the clear vision of the pressure variations throughout width of the impeller. The pressure obtained by the ANSYS CFX is more as compare to the OpenFOAM it this difference is due to the in the technique of the solving and modelling of the wall functions.

5.5 Streamlines

As it has been observed that velocity vector plot is same

for both the cases hence the streamline should be same. Looking at the streamlines for both the cases are same. The differential pressure can only be maintained when the streamlines are sufficiently curved. The equilibrium of forces between the streamline is crucial for recirculation which is not observed in both the cases.

Conclusions

- 1) After studying the results obtained by the ANSYS CFX and OpenFOAM the following points are concluded. ANSYS CFX converges at 350 iterations whereas the OpenFOAM converges at 1000 iterations.
- 2) y^+ Value obtained by ANSYS CFX and OpenFOAM differ by certain value due to usage of scalable wall functions in ANSYS CFX where as in OpenFOAM near wall treatment is not available.
- 3) The velocity vectors in ANSYS CFX and OpenFOAM are same but found recirculation in the case of OpenFOAM.
- 4) The pressure distributions obtain by the ANSYS CFX and OpenFOAM differ at the impeller eye and in volute casing it is nearly same.
- 5) The pressure obtain by the ANSYS CFX is greater than OpenFOAM.
- 6) The head obtain by the ANSYS CFX is less than OpenFOAM by a value of 0.8 m.
- 7) The hydraulic efficiency obtain by the ANSYS CFX is nearer to experimental results than by the OpenFOAM.
- 8) ANSYS CFX predicts good and promising results with experimental results than OpenFOAM.

Future Work

In this work the Reynolds-Averaged Navier-Stokes (RANS) equations supplemented with the SST model are used to model Centrifugal Pump and quite good results were achieved in ANSYS CFX and OpenFOAM. With the implementation of proper boundary conditions and proper Numerical solving techniques OpenFOAM has given results less accurate results than CFX. Overcoming the erroneous steps involved in the present work comparison of Centrifugal pumps of low specific speed are carried out where if OpenFOAM is predicting results very close to the experimental results.

References

- Dixon, S. L., 2008. Fluid Mechanics and Thermodynamics of Turbo-machinery, 5th ed. Elsevier.
- Gulich, J. F., 2008. Centrifugal Pumps. Springer-Verlag Berlin Heidelberg.
- Stepanoff, A. J., 1957. Centrifugal and Axial Flow Pumps. John Wiley & Sons.
- Olivier, P., 2010. Numerical Investigation of Incompressible Turbomachinery Applications using OpenFOAM. Chalmers University of technology.
- Barth, T.J., Jespersen, D.C., 1989. The Design and Application of Upwind Schemes on Unstructured Meshes, AIAA.
- Patankar, S.V., 1980. Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing Corp.
- ANSYS: User's Guide Version 12.1
- Hrvoje, J., 1996. Error Analysis and Estimation for the Finite Volume Method with Applications to Fluid Flows. Imperial College London.