CFD Analysis of Centrifugal Pump using ANSYS

Sahil[1]
[1]CFD Engineer,
Eduhive Engg Consult, Dehradhun,
India

Shubham Singhmar[2]
[2]Student,
Department of Mechanical engineering,
Chandigarh University, India

Dr. Thakur Sanjay Kumar[3]
[3] Associate Professor,
Department of Mechanical engineering,
Darbhanga College of Engineering, Darbhanga

Abstract- **: Centrifugal pumps are a most commonly used in different fields like industries, agriculture and domestic applications. Computational Fluid Dynamics is most commonly used tool for simulation and analysis. 3-D numericalCFD tool is used for simulation of the flow field characteristics inside the turbo machinery. CFD simulation makes itpossible to visualize the flow condition inside centrifugal pump. The present paper describes the head, power,efficiency and to evaluate the pump performance using the ANSYS CFX, a computational fluid dynamicssimulation tool. These simulations of centrifugal pumps are strongly related to cavitation flow phenomena, which mayoccur in either the rotating runner-impeller or the stationary parts of the centrifugal pumps. The numerical simulationcan be used to detect the cavitation in centrifugal pump and to get safe range of operating at different flow rate and rotating speed.**

Keywords: Cfd, Fluid Mechanics, Pump, Ansys

I. INTRODUCTION

Centrifugal pump is a machine that imparts energy to a fluid. This energy can cause a liquid to flow or rise to ahigher level. Centrifugal pump is an extremely simplemachine which consists of two basic parts: The rotaryelement or impeller and the stationary element or casing. The centrifugal pumps are widely used in the worldbecause the pump is robust, effective and inexpensive toproduce. Centrifugal pumps are more economical to own,operate and maintain than other types of pumps. Pumpsoperate via many energy sources, including manualoperation, electricity, engines, or wind power, come inmany sizes, from microscopic for use in medicalapplications to large industrial pumps. Mechanical pumpsserve in a wide range of applications such as pumpingwater from wells, aquarium filtering, pond filtering andaeration, in the car industry for water-cooling and fuel Injection etc.

II. USE OF COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational fluid dynamics (CFD) is the use of computers and numerical methods to solve Problems involving fluid flow. CFD has been successfully applied in many areas of fluid mechanics. These include aerodynamics of cars and aircraft, hydrodynamics of ships, flow through pumps and turbines, combustion and heat transfer chemical engineering. Applications in civil engineering include wind loading, vibration of structures, wind and wave energy, ventilation, fire, explosion hazards, dispersion of pollution, wave loading on coastal and offshore structures, hydraulic structures such as weirs and spillways, sediment transport. More specialist CFD applications include ocean currents, weather forecasting, plasma physics, blood flow and heat transfer around electronic circuitry.

A. Basic Principles of CFD

The approximation of a continuously-varying quantity in terms of values at a finite number of points is called discretisation.

The fundamental elements of any CFD simulation are:

(1) The flow field is discredited; i.e. field variables (, u, v, w, p …) are approximated by their values at a finite number of nodes.

(2) The equations of motion are discretised (approximated in terms of values at nodes):

Control-volume or differential equations (Continuous) → algebraic equations (Discrete)

The main stages in a CFD simulation are:

B. Pre-processing:

– Formulation of the problem (governing equations and boundary conditions);

– Construction of a computational mesh (set of control volumes).

C. Solving:

– Discretisation of the governing equations;

– Solution of the resulting algebraic equations.

D. Post-processing:

– Visualization (graphs and plots of the solution);

– Analysis of results (calculation of derived quantities: forces, flow rates ...)

ISSN: 2278-0181
RACEE - 2015 Conference Proceedings

E. *Forms of the Governing Fluid-Flow Equations*

The equations governing fluid motion are based on fundamental physical principles:

Mass: change of mass = 0

Momentum: change of momentum = force × time

Energy: change of energy = work + heat

In fluid flow these are usually expressed as rate equations; i.e. *rate of change* = …

When applied to a fluid continuum these *conservation* principles may be expressed Mathematically as either:

Integral (i.e. control-volume) equations;

F. *Differential equations.*

Integral (Control-Volume) Approach

This considers how the total amount of some physical quantity (mass, momentum, energy,) is changed within a specified region of space (*control volume*).

For an arbitrary control volume the balance of a physical quantity over an interval of time is

Change = amount in - amount out + amount created

In fluid mechanics this is usually expressed in **rate** form by dividing by the time interval (and transferring the net amount passing through the boundary to the LHS of the equation):

$$\begin{pmatrix}\text{RATE OF CHANGE}\\ \textit{inside } V\end{pmatrix} + \begin{pmatrix}\text{NET FLUX}\\ \textit{out of boundary}\end{pmatrix} = \begin{pmatrix}\text{SOURCE}\\ \textit{inside } V\end{pmatrix}$$

The *flux*, or rate of transport across a surface, is composed of:

Advection – movement with the fluid flow;

Diffusion – net transport by random molecular or turbulent motion.

$$\begin{pmatrix}\text{RATE OF CHANGE}\\ \textit{inside } V\end{pmatrix} + \begin{pmatrix}\text{ADVECTION + DIFFUSION}\\ \textit{through boundary of } V\end{pmatrix} = \begin{pmatrix}\text{SOURCE}\\ \textit{inside } V\end{pmatrix}$$

The important point is that this is a **single, generic equation**, irrespective of whether the physical quantity concerned is mass, momentum, chemical content, etc. Thus, instead of dealing with lots of different equations we can consider the numerical solution of a generic

Scalar-transport equation

Differential Equations

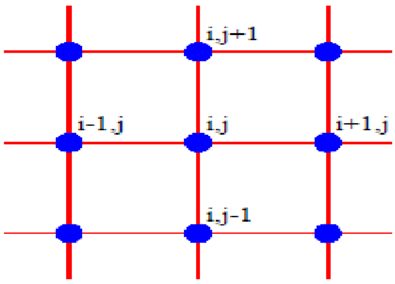
In regions without shocks, interfaces or other discontinuities, the fluid-flow equations can also be written in equivalent differential forms. These describe what is going on at a point rather than over a whole control volume. Mathematically, they can be derived by making the control volumes infinitesimally small.

The Main Discretization Methods

(1) Finite-Difference Method

Discretize the governing **differential** equations; e.g.

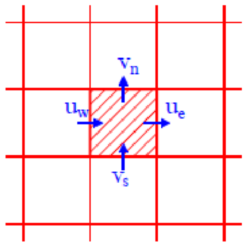
$$0 = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \approx \frac{u_{i+1,j} - u_{i-1,j}}{2\Delta x} + \frac{v_{i,j+1} - v_{i,j-1}}{2\Delta y}$$



(ii) Finite-Volume Method

Discretize the governing **integral** or **control-volume** equations; e.g.

$$\textit{net mass outflow} = (\rho u A)_e - (\rho u A)_w + (\rho v A)_n - (\rho v A)_s = 0$$



(iii) Finite-Element Method

Express the solution as a weighted sum of *shape functions S* (**x**); e.g. for velocity:

$$u(\mathbf{x}) = \sum u_\alpha S_\alpha(\mathbf{x})$$

Substitute into some form of the governing equations and solve for the coefficients u_α

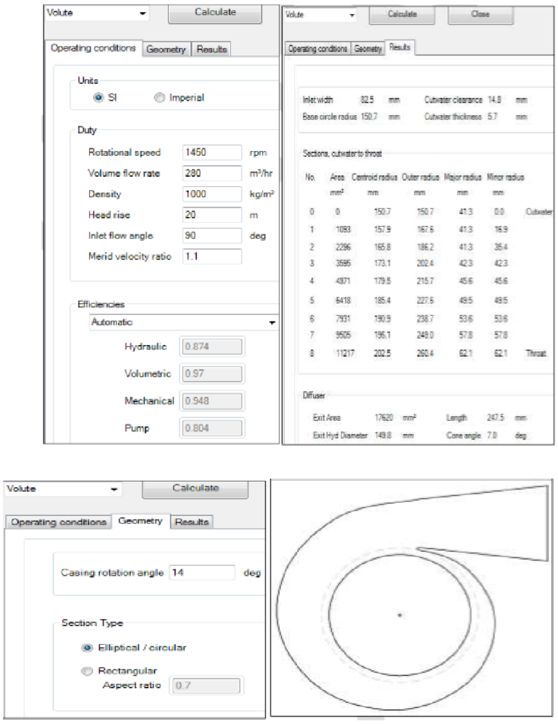
III. CENTRIFUGAL PUMP DESIGN

The design of centrifugal pump is divided in two categories: Impeller Design and Volute Design. The detailed procedure of singlevolute casing and impeller design can be found in different literature; in this paper vista CPD for the design of centrifugal pump isused. The duty parameters required by the pump are assumed to be: 1. Head = 20 m, 2. Flow rate = 280 m/hr, 3. RPM = 1450,4.Density = 1000 Kg/ m^3.

A. impeller design using vista tf v14.5

Input variables are used to give a basic starting point for the pump design. The head, volume flow, rotational speed and otherparameters could be changed to the specific purpose. Various windows show the design parameters, like the angle and thicknessdistribution. The following fig-2 will demonstrate the entire workflow from input values in Vista CPD to final results by Vista designmodule. This way, manipulation of the geometry in BladeGen or BladeEditor will be possible and all the next steps will beautomatically generated and results produced.

B. volute design



IV. GRID GENERATION

Once the pump geometry has been specified and a mesh has been created automatically, where the flow equations need to be solved.

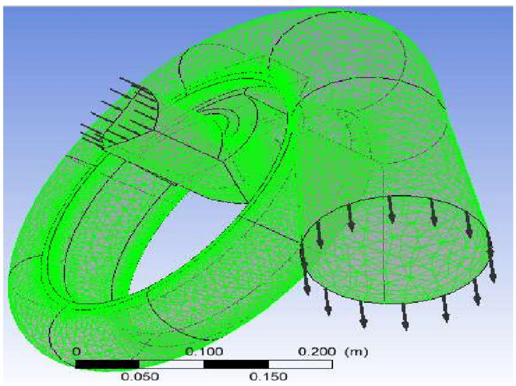


Fig1.: Mesh profile for centrifugal pump.

ANALYSIS SETUP

Design points for a parametric study can be specified using the required duty of the pump in the setup steps:

Input Material: Material is also assigned to the parts of the pump as: Casing and Impeller: Aluminium alloy, Hydraulic Region:Water, Rotating part: Rotating region

Boundary Conditions: Boundary conditions are applied to the inlet and outlet of the pump i.e. 0 pa at inlet, 280 m/hr at outlet, and1450 RPM.

A. solution initialization

Initialization in Ansys CFX is done by providing initial guess values to solve the governing equation so that the flow field variablescan be solved by iteration toward the solution. The default automatic initialization for the velocity and static pressure is used toprovide a start point to the solution.

V. RESULTS

After analysis has been carried out the following results are obtained. The results are taken only when the convergence is obtained forthe solution. As the solution iterated 1000 times and the pump impeller completed a full turn, following results are taken fromdifferent axis and cross-sections.

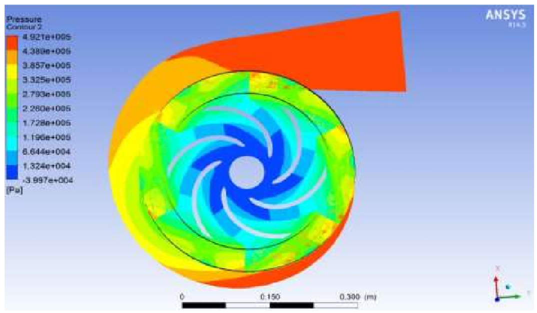


Fig2: Pressure counter flow through the pump in the mid plane view for Q = 36 m^3/kg

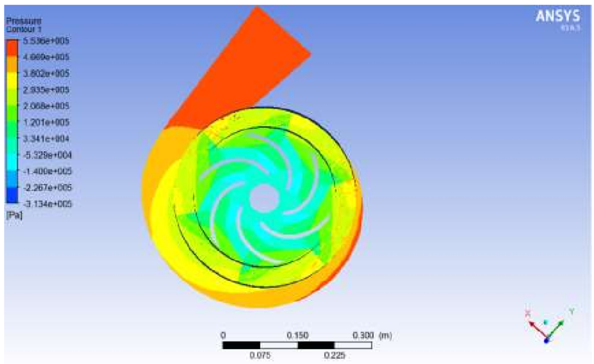


Fig3: Pressure counter flow through the pump in the mid plane view for Q=280m3/kg (At design value of the flow rate)

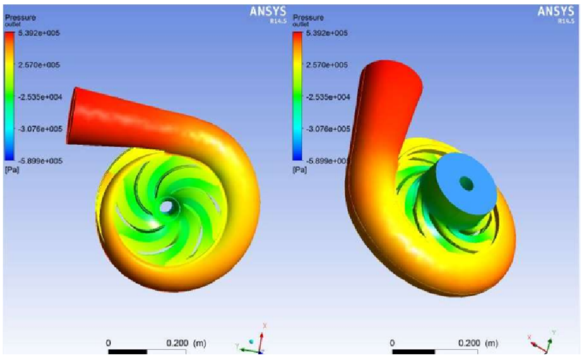


Fig4. Pressure counters flow through the pump

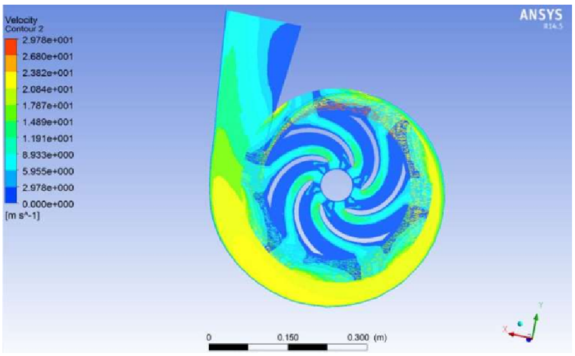


Fig5: Velocity counters flow through the pump in the 3D mid plane view at design value of the flow rate.

VI. CONCLUSION

The design and analysis of the centrifugal pump is done and through analysis we conclude that the deflection on the vane of an impeller is very less and within the design limits and hence the design of the impeller is safe. Experimental methods and past experience are undoubtedly important, but the most effective way to study pump performance is through Computational Fluid Dynamics (CFD).It is found that the design and analysis methods lead to completely very good flow field predictions. This makes the methods useful for general performance prediction. In this way, the design can be optimized to give reduced energy consumption n, lower head loss, prolonged component life and better flexibility of the system, before the prototype is even built.

The efficiency of the centrifugal pump can be improved by increasing the value of surface finish factor. The impeller is the important component of the centrifugal pump which plays a crucial role in determining the efficiency of the centrifugal pump. The fatigue life of shaft is calculated which is infinite. The centrifugal pump operates at high temperature and high pressure, so selection of mechanical seals becomes of utmost importance in this situation. Hence, the design and performance analysis of the pump components is done and efficiency is found out which is satisfactory.

VII. REFERENCES

[1] S.Rajendran and Dr.K.Purushothaman, "Analysis of a centrifugal pump impeller using ANSYS-CFX," International Journal of EngineeringResearch & Technology, Vol. 1, Issue 3, 2012.

[2] S R Shah, S V Jain and V J Lakhera, "CFD based flow analysis of centrifugal pump," Proceedings of the 37th National & 4th InternationalConference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, 2010.

[3] P.UshaShriansC.Syamsundar, "computational analysis on performance of a centrifugal pump impeller," Proceedings of the 37th National& 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, 2010.

[4] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, "Parametric Study of a Centrifugal Pump Impeller by Varying th e OutletBlade Angle," The Open Mechanical Engineering Journal, no 2, 75-83, 2008.

[5] Marco Antonio Rodrigues Cunh and Helcio Francisco Villa Nova, "Cavitation modelling of a centrifugal pump impeller ," 22nd International Congress of Mechanical Engineering, Ribeirao Petro, Sao Paulo, Brazil, 2013.

[6] Mohammed Khudhair Abbas, "cavitation in centrifugal pumps," Diyala Journal of Engineering Sciences, pp. 170-180, 2010.[7] AbdulkadirAman, SileshiKore and Edessa Dribssa, "Flow simulation and performance prediction of centrifugal pumps using cfd -tool," Journal of EEA, Vol. 28, 2011.

[7] Erik Dick, Jan Vierendeels, Sven Serbruyns and John VandeVoorde, "Performance prediction of centrifugal pumps with cfd -tools," Task Quarterly 5, no 4, 579–594, 2001.

[8] S. C. Chaudhari, C. O. Yadav and A. B. Damo, "A comparative study of mix flow pump impeller cfd analysis and experimental data of submersible pump," International Journal of Research in Engineering & Technology, Vol. 1, Issue 3, 57-64, 2013.

[9] D. Somashekar and Dr. H. R. Purushothama, "Numerical Simulation of Cavitation Inception on Radial Flow Pump," IOSR Journal of Mechanical and Civil Engineering, Vol. 1, Issue 5, pp. 21-26, 2012.

[10] Liu Houlin, Wang Yong, Yuan Shouqi, Tan Minggao and Wang Kai, "Effects of Blade Number on Characteristics of Centrifugal Pumps," Chinese journal of mechanical engineering, Vol. 23, 2010.

[11] Myung Jin Kim, Hyun Bae Jin, and Wui Jun Chung, "A Study on Prediction of Cavitation for Centrifugal Pump," World Academy of Science, Engineering and Technology, Vol. 6, 2012.