International Journal of Engineering Technology, Management and Applied Sciences

www.ijetmas.com April 2017, Volume 5 Issue 4, ISSN 2349-4476

Investigation of Main Area of Cavitation in Centrifugal Pump Using Ansys CFX

Nipun P Raval
Institute of Techology,
Nirma University
S-G Highway, Ahmedabad-Gujarat

Tejas N Raval
Institute of Techology,
Nirma University
S-G Highway, Ahmedabad-Gujarat

Anand A Bhatt
Institute of Techology,
Nirma University
S-G Highway, Ahmedabad-Gujarat

ABSTRACT

Centrifugal pumps are mainly used in different fields like agriculture, Industries and household applications. Turbo machine like centrifugal pump faces problems with efficiency and degradation of its material which is caused by cavitation. Cavitation can be described as rapid formation and collapse of vapour cavities formed due to local pressure difference. Flow profile inside centrifugal pump is complex having turbulent and unsteady flow with cavitation phenomenon. Computational Fluid Dynamics (CFD) technique has been helpful to carry out cavitation analysis and performance prediction. This paper presents simulation of cavitation analysis for centrifugal pump impeller in ANSYS-CFX. Here attempt is made to find out the area which are most likely to get cavitation in centrifugal pump impeller.
Keywords-*Centrifugal pump impeller, Cavitation, Ansys CFX, CFD*

INTRODUCTION

Cavitation is the development of vapor bubbles within a liquid which are created when flow dynamics cause the local static pressure to drop below the vapor pressure. The bubbles usually last a short time, collapsing when they encounter higher pressure. The collapse can cause material damage and noise. Cavitation is frequently encountered problem in pumps, inducers, hydraulic turbines, propellers, fuel injectors and other fluid devices subjected to low pressures. Cavitation often causes performance failure and destruction, which can be costly. Because ANSYS CFX software has long been a technology leader in computational fluid dynamics (CFD) for rotating machinery, engineers and designers now look to ANSYS CFX to understand and reduce cavitation. Cavitation is a phase change process. A usual starting point for analyzing cavitation is the Rayleigh-Plesset equation, which describes the growth and collapse of a vapor bubble subjected to a far-field pressure disturbance. Using this equation, it is possible to develop a relationship for the cavitation rate in terms of the fluid properties and the difference between the local pressure and vapor pressure. The particular model implemented in ANSYS CFX software has been tuned to model both the vaporization and condensation stages of cavitation, and it has been validated for a number of working fluids including water and diesel fuel. Using the powerful CFX Expression Language™ feature, users also may implement custom cavitation models.Cavitation can pose significant numerical challenges to a CFD code because of the large density ratio between the liquid and vapor. ANSYS CFX software has long been a leader in the simulation of multiphase flows so it is uniquely suited to address these challenges. The cavitation model is implemented as a homogeneous multiphase model, in which a zero slip velocity between liquid and vapor is assumed. The global continuity equation is expressed in volumetric form to minimize robustness problems at the liquid-vapor interface. With this formulation, the cavitation source appears as a volume source term that can be linearized in terms of pressure; the resulting equation set can be solved efficiently using ANSYS CFX software's unique implicit coupled multigrid solver.

MODEL

Here problem consist of a five blade centrifugal pump operating at 2160 rpm. The working fluid is water and flow is assumed to be steady and incompressible. Due to rotational periodicity a single blade passage will be modeled. Initially flow field is solved without cavitation later it is turned on. Fig. 1 shows full impeller profile. Fig. 2 shows blade passage

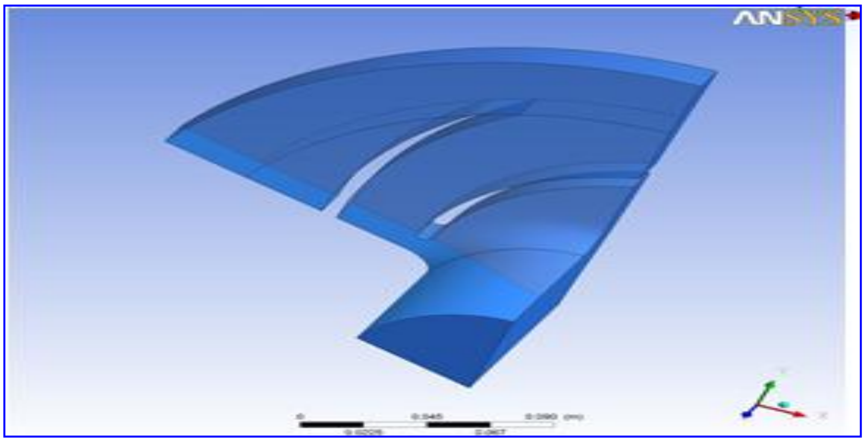

Figure.1 Full impeller profile

Creating working fluid and setting up the fluid domain in Ansys CFX

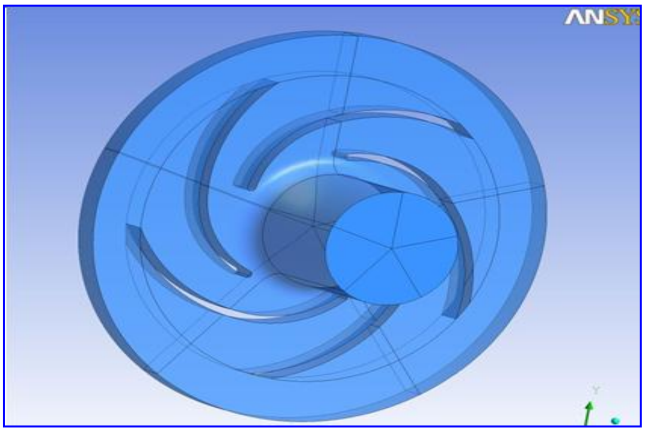

Figure 2 Blade passage

In basic settings in CFX pre in material properties
water with density 1000 kg/m3 is selected. Dynamic viscosity is kept as 0.001kg / m-s. Molar mass is set as 18.02 kg/ kmol. Specific heat capacity is taken as 4181.7 J/kg –K. Reference temperature and reference pressure is taken as 250 C and 1 atmosphere respectively. Reference value of both specific enthalpy and specific entropy is kept as zero. Thermal conductivity value is set as 0.6069 W/m-K. To perform cavitation analysis two fluid domain named 'Water' and 'Water vapour at 25 C' are crated under default fluid domain in CFX pre.
Reference pressure is set to 0 Pa. It is required to keep domain motion rotating. Rotational speed of impeller is taken as 2160 rev/min. Here alternate rotational model is used to avoid false swirl which could occur when a significant amount of fluid is flowing in the axial direction. It is necessary to set axis definition to global Z. Homogeneous model in fluid model tab is selected. Heat transfer mode is kept isothermal at 250 C. Turbulence option is selected as shear stress transport.

Inlet and Outlet Boundary Conditions

Boundary condition named Inlet is created. In CFX Pre in basic setting tab boundary type is set Inlet. Location is set as inlet of impeller. The frame type is kept stationary. In boundary detail tab mass and momentum with a Normal speed of 7.0455 m/s is specified. In fluid value tab volume fraction for water is set as 1 and for water vapour volume fraction is kept as 0. After creating outlet boundary condition named Outlet boundary type is kept to opening in basic setting tab. In outlet too boundary condition frame is set to stationary. In outlet mass and momentum is specified using Entrainment, and relative pressure of 600000 Pa is entered.

In outlet pressure option is enabled and set to opening pressure. Turbulence is set to zero gradient. For outlet boundary condition volume fraction for water liquid and water vapour is kept as 1 and 0 respectively. Due to symmetry of the shape periodic interface is preferred. Interface Models option is set to rotational periodicity. Global Z axis is selected. By default, all walls in a rotating domain rotate with the rotating reference frame. Since this wall is stationary in the absolute frame it must be counter rotating in the rotating frame.

Initialization and solver control

Water liquid volume fraction is set to 1 with automatic option in fluid setting form. Vapour volume fraction is kept as 0. A commonly used timescale in turbo machinery is 1/omega , where omega is the rotation rate in radians per second. In this case , 2/omega is used to achieve faster convergence. 1/ (pi *2160[min^-1]) is used in physical timescale. Residual target is set to 1e-5. To speed up the convergence multiphase control turned on and also volume fraction coupling is turned on. Fig.3 shows convergence in CFX.

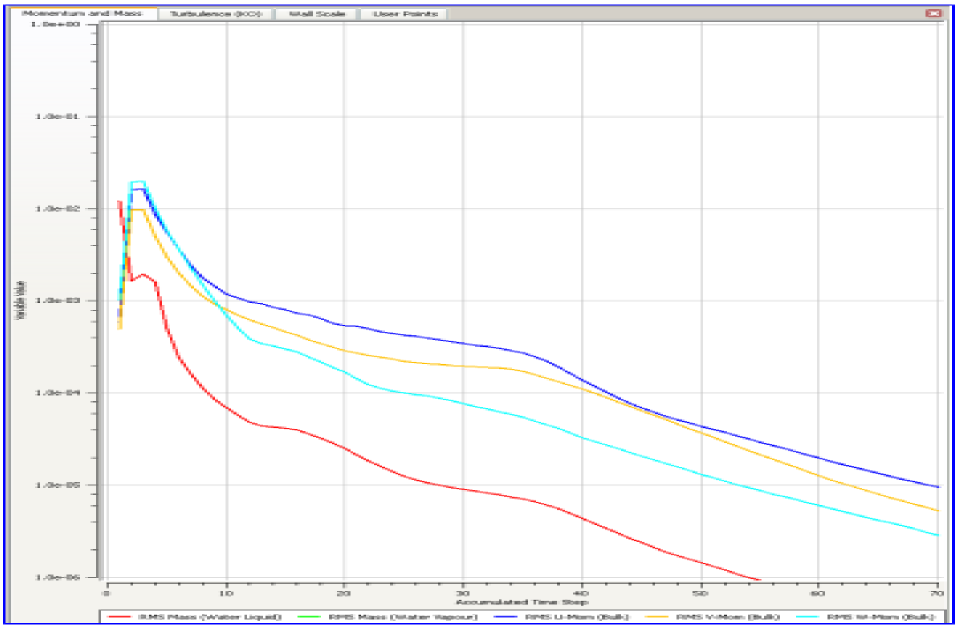

Figure 3 Convergence in CFX

Post Processing

Blades are very critical component of impeller and performance of impeller largely depends on blades so here contour is inserted selecting the blade region. Absolute pressure is set and range is kept as global. Hub and shroud are also important part of impeller and contour using variable absolute pressure is set over local range for bot shroud and hub. The minimum pressure is above the saturation pressure of 2650 Pa for water here as depict from Figure 4, Figure 5 and Figure 6.

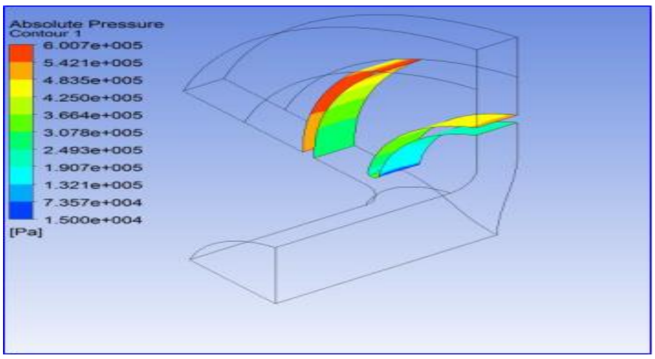

Figure 4 Contour of Blade Pressure

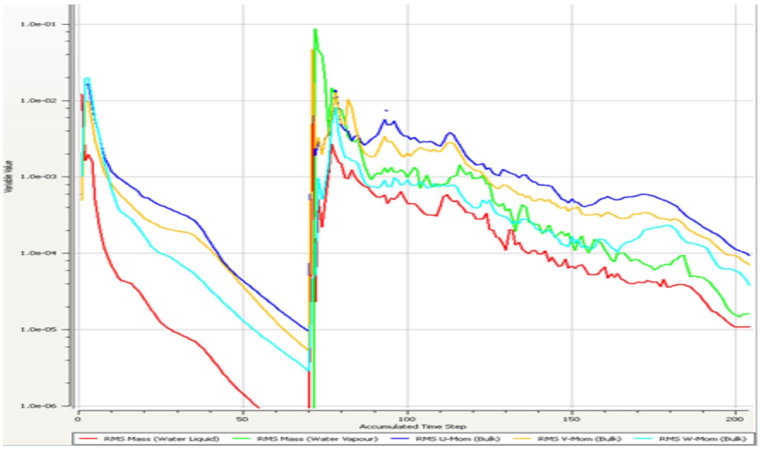

Figure 5 Contour of Hub Pressure

Another part of simulation includes cavitation analysis. Some of the physic modification are required for cavitation analysis

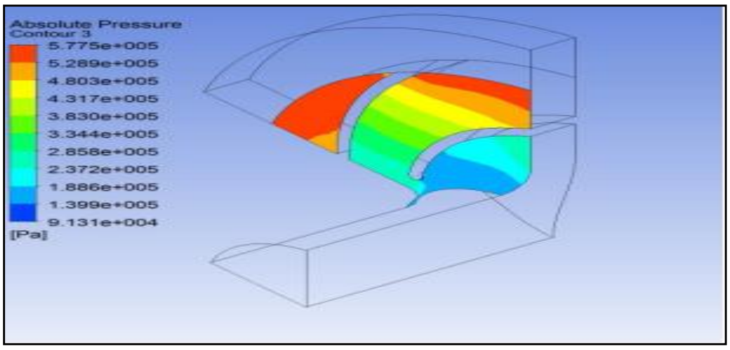

Figure 5 Contour of shroud Pressure

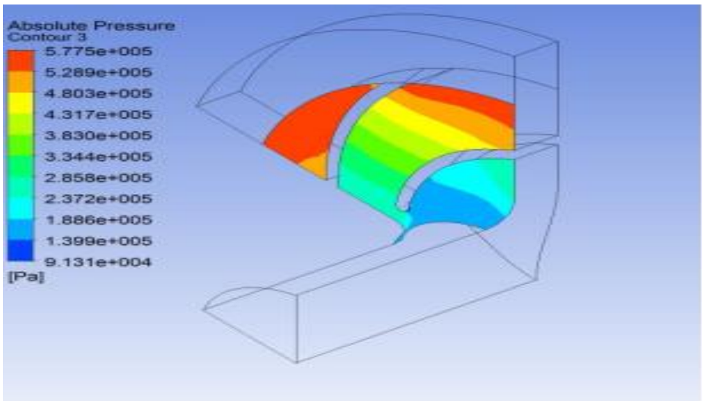

Figure 6 Convergence due to Cavitation

such as in fluid pair model , mass transfer is set to cavitation. Rayleigh Plesset option is taken and mean nucleation site diameter is set to 2e-6, this is reasonable value. Saturation pressure is kept as 2650 Pa. Relative pressure is taken as 300000 Pa. Most cavitation solution should be performed by turning cavitation on and then successively lowering system pressure over several runs to induce cavitation gradually. This approach is suitable for modeling almost all industrial cases. Here in cavitation analysis in solver maximum iterations is kept as 150 and residual target is kept as 1e-4. There is a significant spike in residuals as shown in Fig. 6, in part due to the outlet pressure difference, but also due to the fact that absolute pressure is low enough to induce cavitation.

Results of cavitation analysis show that minimum pressure is equal to saturation pressure specified earlier. This suggests that some cavitation has occurred as shown in Fig. 7 and Fig. 8.

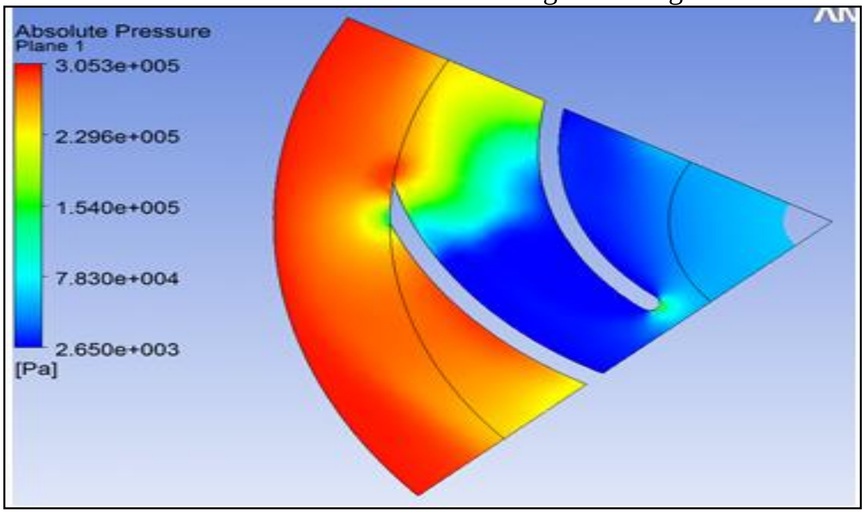

Figure 7 Occurrence of Cavitation

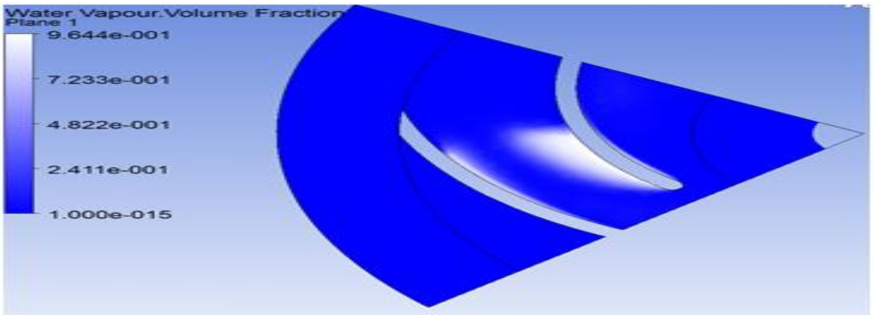

Figure 8 Water vovapour volume fraction

Further processing includes crating volume using isovolume method and variable is set to water vapour volume fraction. 360 degree view of model make the cavitation phenomenon more clear.

SUMMARY

The main area of cavitation exists between suction side of the blade and the shroud in this geometry. A secondary area of cavitation is just behind the leading edge of the blade on the pressure side. Thus special care is required suction side of blade as well as shroud. It is clear from figure 9 that cavitation has devastating effect on impeller and specially on blade geometry. Certain precautionary steps like maintaining net positive suction head as per the requirement can avoid cavitation. This method of cavitation analysis is well suited for industrial purpose also. Further we can also calculate

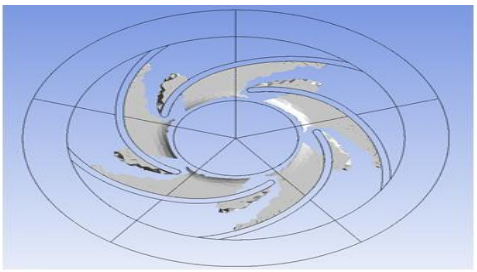

Figure 9 Wireframe model of impeller after cavitation

torque on the blade using function calculator. Velocity vectors can also be plotted stationary frame. Mass flow rate and total pressure drop from inlet to outlet can also be calculated using this simulation.

REFERENCES

[1] Kim, M.J., Jin, H.B, and Chun, W.J., " A study on prediction of cavitation for centrifugal pump", International Journal of Mechanical Aerospace, Industrial, Mechatronics and Manufacturing engineering, 2012, Vol: 60 ; 2720-2725

[2] A.J.Stepanof , Centrifugal and axial flow pumps theory design and application , 2nd ed, John wilay and sons, New York.

[3] Mihaeloff A.K and Malish, V.V 1971,, Construction and calculation of high pressure centrifugal pumps,1st ed, Moscow publishing house ,.

[4] Turton R.K, Rotodynamic Pump Design ,2nd ed, Cambridge University Press, New York.

[5] Lomakin, A.A.,1966, Centrifugal and Axial flow pumps,1st ed, Moscow Publishing House

[6] Tuzson, J. Centrifugal Pump Design., 2nd ed ,John Wiley and Sons Inc., New York.

[7] Evans, J. , A brief Introduction to centrifugal pump[Online]. Avilable: http:// www.pacificliquid.com.

[8] Chaurette, J., Tutorial on centrifugal pump system [Online]. Avilable: http:// www.lightmypump.com.

[9] Platt, M.J, Cook, C." 2002 Thermo Mechanical Turbo pump Design and Analysistools", 2nd Modeling simulation subcommittee,destin,florida..

[10] Anagnostopoulos , J.S. " A fast numerical method for flow analysis and blade design in centrifugal pump impellers", 2009 Computers Fluids 38

[11] Jafarzadeh ,] B. , Hajari , A., Alishahi , M.M., Akbari, (2011)"The flow simulation of a low-specific-speed high-speed centrifugal pump", Applied Mathematical Modelling 35.

[12] ANSYS CFX-Manual 2006, Published by, ANSYS CFX, Release 11.0, ANSYS, Inc