Computational Fluid Dynamics (CFD) of Centrifugal Pump to Study the Cavitation Effect

[1]R. M. Pande, [2]S. U. Kandharkar, [3]R. B. Patthe, [4]V.M.Nandedkar, [5]V.B. Tungikar

[1,2]UG student [3]Assistant Professor, [4] Professor, [5] Professor and Head

[1-5]Dept. of Prod.Engg,S.G.G.S.I.E.&T. Nanded, Maharashtra. (India)

Email: [1]panderohan@sggs.ac.in, [2]kandharkarsharad@sggs.ac.in, [3]patthe.ramdas7@gmail.com

Abstract:The centrifugal pump iswidely used pump type in industries. The centrifugal pump has been used inthe fields of petroleum, chemical industry, aviation, pharmaceutical industry, metallurgy. Centrifugal pump creates an increase in pressure by transferring mechanical energy from the motor to the fluid through the rotating impeller. The fluid flows from the inlet to the impeller centre and discharged along its blades. The centrifugal force increases the fluid velocity and consequently the kinetic energy is transformed to pressure.A pump in rare situations is observed to present a partial or total failure caused by manufacturing defects. In real life 90–95% of the failures experienced by the equipment may be attributed to one or several possible causes: aeration, cavitation, contamination, excess pressure, excessive temperature or inadequate viscosity of the fluid.

A general three-dimensional simulation of turbulent fluid flow is presented to predict velocity and pressure fields for a centrifugal pump. ANSYS^{TM} 15.0 CFX is used to solve the governing equations of the flow field.Also the objective of this work is to characterize the behaviour of a centrifugal pump.

Keywords: centrifugal pump, CFD, cavitation.

I. INTRODUCTION

Centrifugal pumps are prevalent for many different applications in the industrial and other sectors. Nevertheless, their design and performance prediction process is still a difficult task, mainly due to the great number of free geometric parameters involved. On the other hand the significant cost and time of the trial-and-error process by constructing and testing physical prototypes reduces the profit margins of the pump manufacturers. For this reason, CFD analysis is currently being used in hydrodynamic design for many different pump types [1].

Numerical simulations can provide reasonably accurate information on the fluid behaviour in the machine, and thus help the engineer to obtain a thorough performance evaluation of a particular design.In the current study the effect of turbulence models (Realizable k–e model) on the flow field and cavitation phenomenon of a high-speed centrifugal pump has been carried out. Numerical simulation is performed by using commercial ANSYS^{TM}15.0 CFX package assuming steady flow.

II. LITERATURE REVIEW

All pumps require well-developed inlet flow to meet their potential. A pump may not perform or be less reliable as expected due to a faulty suction piping layout such as a close-coupled elbow on the inlet flange. When poorly developed flow enters the pump impeller, it strikes the vanes and is unable to follow the impeller passage. The liquid then separates from the vanes causing mechanical problems due to cavitation and vibration.Performance problems due to turbulence and poor filling of the impeller. This results in premature seal, bearing and impeller failure, high maintenance costs, high power consumption, and less-than-specified head and/or flow.

JieJina Ying F ana and Wei Hana Jiaxin Hu[2]presented work based on Design and Analysis on Hydraulic Model of The Ultra -low Specific-Speed Centrifugal Pump.With the development of the petrochemical industry and the space technology, lower ultra-high speed centrifugal pump is gradually developed, with its speedmuch higher and the flow rate much lower than the low-specific speed centrifugal pump. Blade number selection of impeller has certain effect for high speed centrifugal pump head, efficiency and cavitation. The main function of the volute is to collect the high speed fluid flow from the impeller, reduce the speed, and convert the kinetic energy into pressure energy.

RymaAchouri ,OmeimaNouicer , HatemMhiri , Philippe Bournot [3] have worked to investigate the probable cause for cracks observed on vertical centrifugal pump. The numerical modelling of the flow through the vertical centrifugal pumpis considered. As part of the numerical simulation, hypothesis of an unsteady flow and an incompressible fluid (seawater) is assumed as a turbulence model, the choice is the k–e standard model. This model is essentially a high Reynolds number model and assumes the existence of isotropic turbulence and the spectral equilibrium.

R. Barrio a, J. Fernandez b, E. Blanco a, J. Parrondo[4] worked out on estimation of radial load in centrifugal pumps using computational fluid dynamics. CFD codes can be used to solve the Reynolds-averaged Navier–Stokes equations (RANS) with the purpose of estimating the performance characteristics of the machine at the design stage. The commercial code Fluent was used to solve the full 3DReynolds-averagedNavier–Stokes equations for unsteady flows (3d-urans) by the finite volume method.

Jose Caridad , Miguel Asuaje , Frank Kenyery , Andrés Tremante , Orlando Aguillón[5] carried out the experimental work on “Characterization of a centrifugal pump impeller under two-phase flow conditions”, This study presents the results of numerical simulations carried out in a centrifugal pump impeller of an Electrical Submersible Pump conveying an air–water mixture.The analysis of a bubble motion through a rotating field is of great importance for numerous engineering problems. Cavitation, boiling, heat and mass transfer applications are good examples.

When a pure liquid is subjected to a pressure below its vapour pressure (at a given, uniform temperature) it is said to be under tension. If there is no vapour present, that state of tension can be stable and, like a solid, a pure liquid can sustain very large negative pressures without rupturing and forming vapour voids. This is not as surprising as it may seem when one considers that a liquid is very similar to a solid in terms of its density and intermolecular forces. The process of vapour bubble formation by this mechanism is called homogeneous nucleation and the corresponding limiting tension can be predicted by kinetic theory. The term boiling refers to a process in which the thermodynamic state of a liquid is changed by heating it at relatively constant pressure. The formation of bubbles resulting from depressurization at relatively constant temperature is called cavitation [1].

III. OBJECTIVES

The objectives for CFD analysis of centrifugal pump are listed below:

- To study the effect of cavitation phenomenon in a centrifugal pump, this involves the use of a rotational domain.
- To evaluatepressure distribution at blade and shroud region of the centrifugal pump.

IV. PROBLEM STATEMENT

Since the fluid surrounding the impeller rotates around the axis of the pump the equations must be organized in two reference frames, stationary and rotating reference frames. To accomplish this, the Multiple Reference Frame (MRF) model [7] has been used. In this approach, the governing equations are set in a rotating reference frame and centrifugal forces added as source terms. The key parameters in the problem statement are described in the following manner:

- The problem consists 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 modelled.

ANSYS^{TM}15.0 CFX provides set of features for solving problems in which fluid rotates around an axis, such as flows inside turbo-machineries in different methods. Some of these methods include multiple reference frames (MRF), mixing plane and sliding mesh models [7]. Each method has a different accuracy and computational expenses. The first and second models are appropriate for steady flows and for cases in which the interactions between rotor and stator are negligible. For instance, for a pump with bladeless stators the MRF can be used as a suitable approach. The sliding mesh model is appropriate where the interaction between rotor and stators is noticeable and the steadiness of problem is supposed to be reproduced.

V. METHODOLOGY

During CFD analysis, mainly fluid and flow properties are found to affect the system performance. The details for each step in CFD analysis are given below:

A. Pre-processing phase

During this phase, Creation of the CAD model using CAD Software like Pro-E, CATIA and fluid and flow domain of the problem have to be defined in ANSYS^{TM}15.0 CFX. The symmetry effect of the CFD analysis is used to simplify the geometry.Reference Pressure, Angular Velocity to 2160 “rev/min”, Rotating motion, and two fluids namely water and water vapour, relative pressure of 600 “KPa” and moving interface have been specified. Polyhedral solid elements are used for CFD analysis which has three DOF.

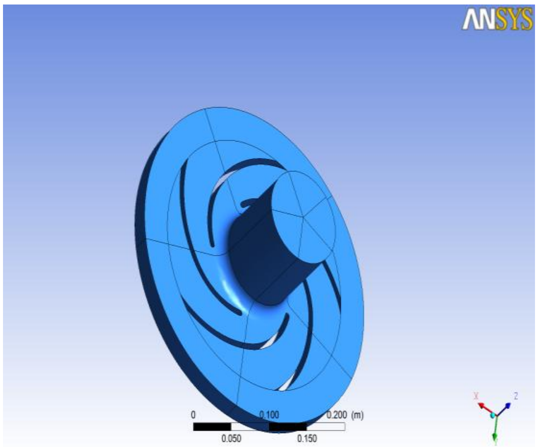

Fig.1CAD model created inPro-E

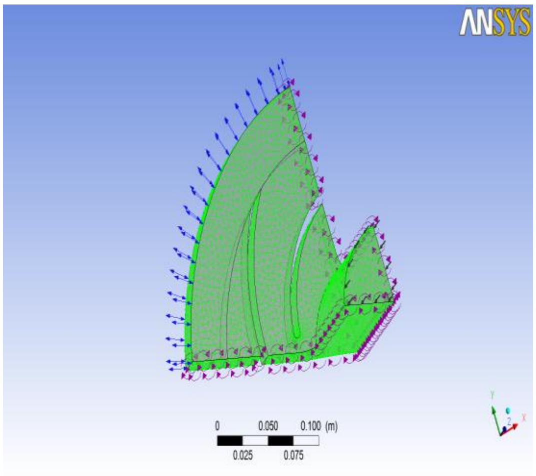

Fig.2Loading and Boundary conditions[7]

B. Processing phase

Following turbulence model are used to plot pressure distribution of the centrifugal pump

i) Standard k-ε model- used mostly for high Reynolds no. / Turbulent flows.

ii) RNG k-ε model-mostly used for rapidly strained flows.

These models have following characteristics:

- Consistent with the physics of turbulent flows
- Accurately predict separating rate of both planer and round jets
- provide superior performance for flow involving rotation

In order to calculate the flow field a commercial CFD code, FLUENTis used. The governing integral equations for the conservation of mass, momentum and when appropriate, energy and other scalars such as turbulence were solved.

C. Post-processing phase

During this phase of CFD analysis, following things need done:

- Interpretation of the results
- Pressure Distribution plot at blade and shroud region of the centrifugal pump

VI. RESULTS AND DISCUSSION

In the present study, the intent is to investigate the effect of five blades on the pump cavitation. Considering various turbulence models, three known turbulence models of standard k–e, RNG k–e and RSM are utilized at a constant blade number of 5 for the pump. In order to show the results in a more practical order non-dimensional parameters defined in reference are used and for the sake generality the data are reported in non-dimensional form.

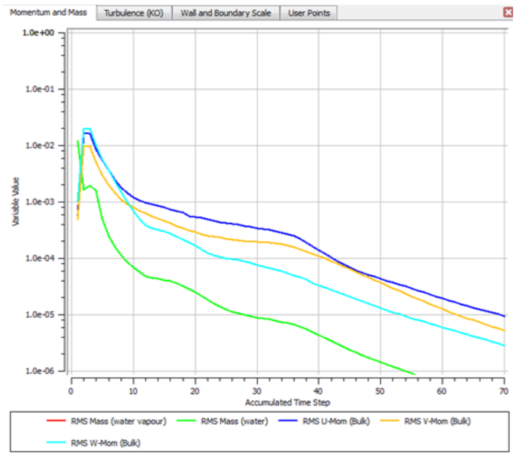

Fig.3 Scaled Residuals for without Cavitation model[7]

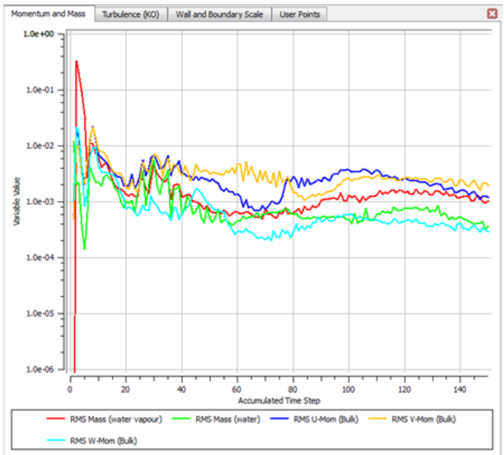

Fig.4 Scaled Residuals for Cavitation model[7]

From above figures, it is observed that less variationsare found in scaled residuals for without cavitation model while more variations in scaled residuals for cavitation model.Aside from the time per iteration, the choice of turbulence model can affect the ability of ANSYS^{TM}15.0 CFXto obtain a converged solution. The RSM may take more iteration to converge than the k–e model due to the strong coupling between the Reynolds stresses and the mean flow.

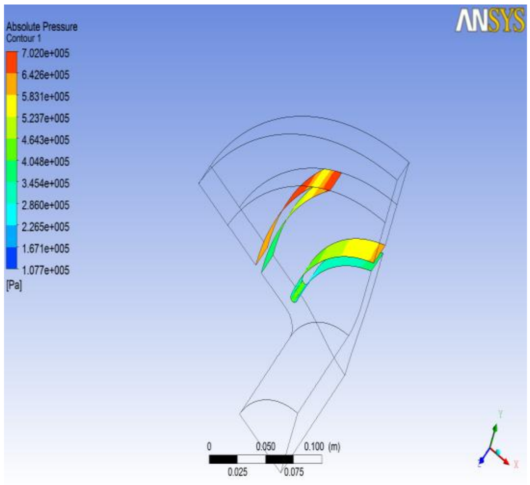

Fig.5Pressure distribution at blade for without Cavitation model[7]

- After setting the some physical modifications for cavitation model, mass transfer to cavitation and saturation pressure of 2650 “Pa” and outlet relative pressure to 300 “KPa” have to be applied. Most cavitation solutions should be performed by turning cavitation on and then successively lowering the system pressure over several runs to more gradually induce cavitation.

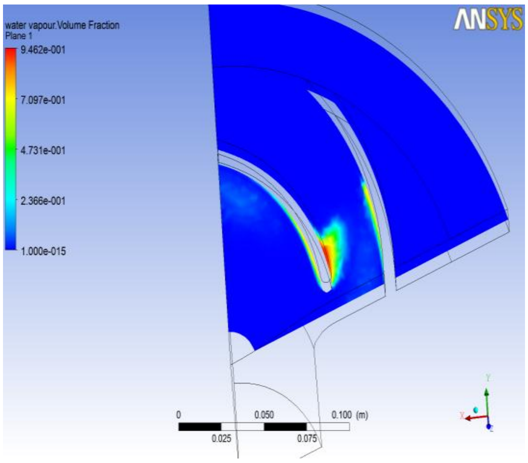

Fig.6 Cavitation induced at the pressure drop region on the blade of the centrifugal pump[7]

VII. CONCLUSIONS

This study describes a FVM method to examine the pump cavitation at the pressure drop region on the blade. The method is based on an analysis of the pressure field in the pump suction and discharge, and a simple pump model. It allows one to isolate the pump-generated pressure pulsations from the pressure measurements local to the pump.

Following conclusions can be drawn from the CFD analysis:

- There is a significant spike in residuals, in part due to the outlet pressure difference
- Absolute pressure is low enough to induce cavitation.

REFERENCES

[1] H.K. Versteeg, W. Malalasekera,“An Introduction to Computational Fluid Dynamics”, Prentice Hall, Loughborough, 1995.

[2] Jie Jin Ying Fan, Wei Han Jiaxin Hu, “Design and Analysis on Hydraulic Model of The Ultra - low Specific-speed Centrifugal Pump”, International Conference on Advances in Computational Modeling and Simulation,Vol.31, pp.110–114, 2012.

[3] RymaAchouri , OmeimaNouicer , HatemMhiri, Philippe Bournot, “Probable cause analysis of cracks observed on vertical centrifugal pump” , Engineering Failure Analysis Vol.29, pp.1–11, 2013.

[4] R. Barrio, J. Fernandez, E. Blanco, J. Parrondo, “Estimation of radial load in centrifugal pumps using computational fluid dynamics”, European Journal of Mechanics B/Fluids Vol.30, pp.316–324,2011

[5] Jose Caridad , Miguel Asuaje , Frank Kenyery ,Andres Tremante, Orlando Aguillón , “Characterization of a centrifugal pump impeller under two-phase flow conditions”, Journal of Petroleum Science and Engineering ,Vol.63,pp.18–22, 2012.

[6] B. Jafarzadeh, A. Hajari, M.M. Alishahi , M.H, Akbari,“The flow simulation of a low-specific-speed high-speed centrifugal pump”, Journal of Applied Matheatical Modelling 35,242–249,2011.

[7] ANSYS^{TM} Fluent 15.0 Theory Guide

◈◈◈