Proceedings of Conference on Advances on Trends in Engineering Projects (NCTEP-2019)
In Association with Novateur Publications IJIERT-ISSN No: 2394-3696
ISBN. No. 978-93-87901-03-2
February, 15th and 16th, 2019

PAPER ID: ME126

NUMERICAL AND CAVITATION ANALYSIS OF A CENTRIFUGAL PUMP

N.D Dhote
Assistant Professor, Department of Mechanical Engineering, College of Engineering, Pune-411005
ndd.mech@coep.ac.in

Yogesh Waykar
M. Tech (Design Engineering), College of Engineering, Pune-411005.
waykaryr17.mech@coep.ac.in

M.P Khond
Associate Professor, Department of Mechanical Engineering, College of Engineering, Pune-411005
mpk.mech@coep.ac.in

***ABSTRACT*- This paper aims at numerically predicting the flow behavior of the pump. Further validation of the available experimental results and inspection of the effect of change in variation of parameters along with cavitation phenomenon was numerically studied and simulated. The said objective is accomplished by creating the three-dimensional geometry of the pump in CAD package. Hybrid discretization was carried out and the results are analyzed and numerically obtained by using ANSYS Fluent R15. The effect of variation of rotational speed, discharge, and change in number of blades individually on the performance of the pump was also studied to get clear idea about the performance of the pump and the behavior of the fluid inside the pump. The cavitation phenomenon is also numerically modelled on the three-dimensional impeller geometry to analyze inception and progress of cavitation bubbles.**

Keywords- Cavitation, Centrifugal Pump, CFD, Discretization, Numerical analysis, Vapour volume fraction.

I. INTRODUCTION

Centrifugal pumps are used extensively for pumping water over short to medium distance through pipeline where the requirements of head and discharge are moderate. In today's competitive and sophisticated technology, centrifugal pump is more widely used than any other applications because of the advantages of centrifugal pump like low initial cost, high efficiency, uniform discharge and continuous flow. Centrifugal pumps are widely used for irrigation, agriculture, water supply plants, steam power plants, sewage, oil refineries, chemical plants, hydraulic power service, food processing factories and mines. Moreover, they are also used extensively in the chemical industry because of their suitability in practically any service and are mostly used in many applications such as water pumping project, domestic water raising, industrial waste water removal, raising water from tube wells to the fields. This paper targets numerical validation of available experimental data and prediction of cavitation inception using ANSYS Fluent R15. The problem of cavitation requires the prompt solution because some of the negative consequences including the slowdown in performance, increase in noise and vibration will make the pump work abnormally and greatly shorten the pump service life which will eventually lead the machine being scrapped.

It is proposed to carry out the computational analysis of a commercial centrifugal pump of design capacity Q = 60 m^3/h, whose experimental data is available. Its specifications are as follows:

Table 1.1 Specifications of Pump

Number of blades, Z	*8*
RPM	2900
Impeller diameter, mm	214
Inlet blade angle	56^0
Outlet blade angle	18^0

The pump specifications at best efficiency point are: Flow rate Q = 60 m^3/h, head H = 56 m, rotational speed N = 2900 RPM, η = 72 %.

Cavitation phenomenon mainly occurs near the impeller eye of the pump due to the low pressure created in that zone. This low pressure can go below the vapor pressure of the liquid which will cause boiling of the liquid. In this case water is used as liquid which as a vapor pressure of 3540 Pa at room temperature. For this reason it is decided to model the impeller differently to analyze cavitation. This also has the added advantage of saving the computational time which otherwise would be used for modelling and discretizing the inlet pipe, casing and delivery pipe.

II. GEOMETRY

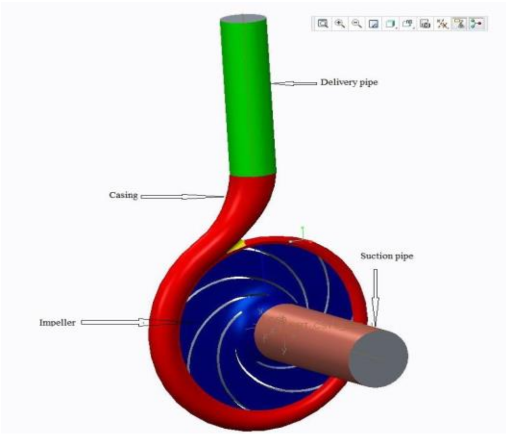

Fig 2.1 Three-dimensional geometry of pump

The problem under consideration is to be modeled first before starting the actual simulation Fig. 2.1 shows the geometry used for analysis purpose. It consists of an inlet pipe, a rotating impeller, casing and a delivery pipe. The geometry is made in Creo Parametric 2.0.

Fig. 2.2 shows the impeller under consideration for cavitation purpose. It is modelled and discretized separately.

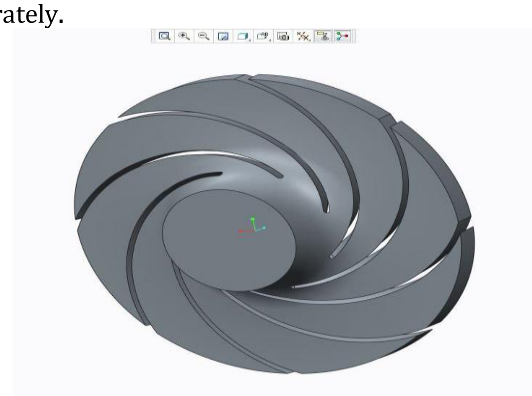

Fig. 2.2 Impeller geometry

III. DISCRETIZATION

After geometric modelling of the pump, it is required to discretize the assembly using suitable meshing module. Discretization is the process of dividing the domain into small number of elements (two-dimensional) or volumes (three-dimensional), over which the governing equations are solved by the solver. At the time of importing geometry, some data loss may take place (example: gaps may create, surfaces may overlap, edges may distort) that should be corrected before creating mesh, otherwise required mesh quality will not obtain. Normally the grid is constructed from lower topologies to higher topologies, i.e. first the edges are meshed, then the faces and finally the volumes As our domain is three-dimensional, the domain is discretized into finite volumes. Hypermesh13.0 is used for this purpose and the domain is obtained as shown in Fig. 3.1. For better solution one may need to refine the mesh at certain areas of the geometry where the gradient of the field variable whose solution is sought is high. But for a highly refined mesh, where the number of cells per unit area is maximum the CPU time required will be relatively large.

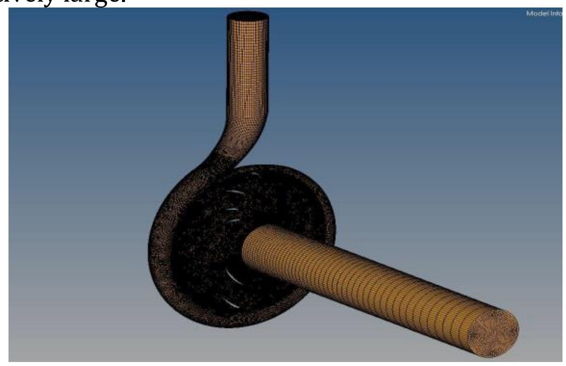

Fig. 3.1 Computational domain of Pump

An important aspect of meshing is determining its quality, which drastically affects the solution. The mesh statistics of various parts of the domain are given in Table 3.1.

Table 3.1 Mesh Statistics

Part	No. of nodes	No. of elements	Skewness
Inlet pipe	128485	311630	0.4
Impeller	372028	1233637	0.84
Casing	95878	321340	0.72
Delivery Pipe	74763	136487	0.5

Once the geometry is defined and the appropriate model is chosen, the boundary conditions have to be specified. Boundaries are typical inlets, outlet, walls, symmetry planes, periodic planes or an axis for axis symmetrical computations. Also for applying the boundary conditions, named selections are assigned to different surfaces. In this case the inlet pipe surface is assigned velocity inlet boundary condition, while the outlet surface of the delivery pipe has been given outflow boundary condition. The main characteristic of the outflow boundary condition is that it is used when the pressure and velocity of the fluid at that particular boundary are not known prior to the calculation. Hence it is a handy element for use.

IV. VALIDATION

Validation is nothing but providing the evidences of results which confirms the working methodology is right and the boundary conditions used for obtaining the results are correct. A numerical solution always requires validation either with the experimental data or against physical laws. Since the results are evaluated computationally the solution methodology is validated with the experimental data available. Results of above mentioned centrifugal pump are published therefore, used for the validation purpose.

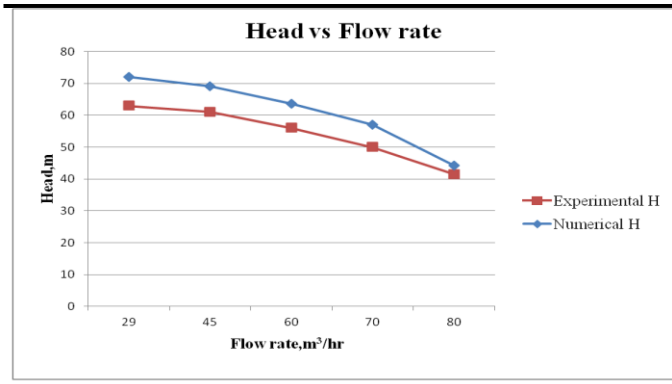

Fig. 4.1: Head vs Flow rate

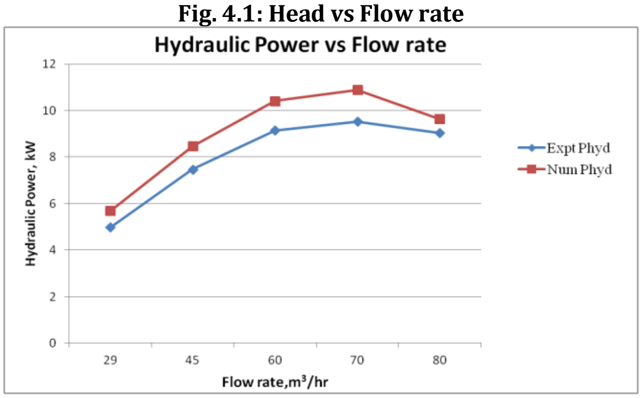

Fig. 4.2 Hydraulic power vs Flow rate

Fig. 4.1 and Fig.4.2 show us the graphs of experimental and numerical head obtained and the respective hydraulic power versus the flow rate in the pump. The maximum error obtained in between results was 10% which is fairly acceptable. It can be seen that the trend followed by experimental data is also followed by numerical results obtained.

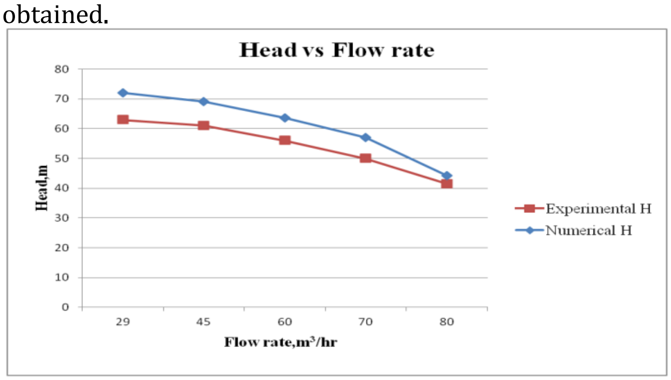

Fig. 4.3 Input power vs Flow rate

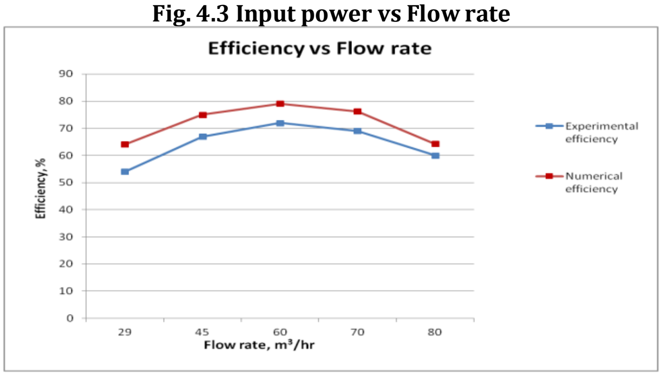

Fig. 4.4 Hydraulic efficiency vs Flow rate

The above figures show us the variation of input power and hydraulic efficiency versus flow rate- both experimental and numerical. Similar to the previous results, the profile of curves obtained is similar in experimental and numerical cases.

V. CAVITATION

The Bernoulli's equation can be applied to the inlet pipe, i.e. to the velocity inlet surface and the impeller eye.

$$\frac{P_i}{\rho g} + \frac{V_i^{\,2}}{2g} + z = \frac{P_{atm}}{\rho g} + h_f$$

Where,
P_i= Inlet pressure at impeller eye.
V_i=Inlet velocity at impeller eye.
z= Height of inlet above fluid surface.
For the left hand side energy to remain constant, as the velocity inlet increases, the static pressure P_i decreases. When this static pressure P_i falls below the vapor pressure P_v of water, it will start to boil, or in other words, cavitation will occur.
Q/Q_D ratio is the ratio of flow rate to the design flow rate, i.e. at the best efficiency point. It is common practice to plot curves against this Q/Q_D ratio.

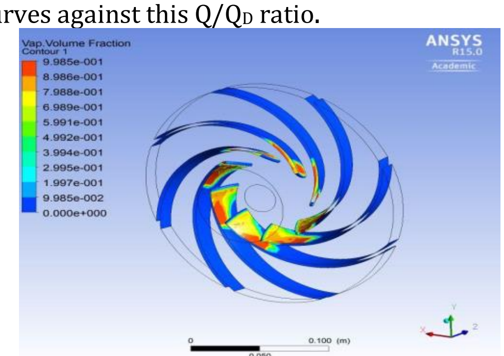

Fig. 5.1 Maximum vapor volume fraction on blades for Q/Q_D =1.33

Fig. 5.1 shows the case where inlet velocity is maximum i.e. for Q/Q_D=1.33. The static pressure falls to a minimum value which is below the vapour pressure of water. Therefore, maximum cavitation is observed in this case. The maximum value of vapour volume fraction obtained here is 0.99.

Fig. 5.2 Maximum vapor volume fraction on blades for Q/Q_D =1.16

Fig.5.2 shows the case where the inlet velocity is maximum i.e. for Q/Q_D =1.16.

As we go on decreasing the inlet velocity gradually the static pressure will go on increasing to the point where it will no longer be lower than the vapour pressure. At this point boiling will not occur and cavitation is not detected and Fig.5.3 shows such phenomenon where Q/Q_D =1.16.

For cases with lower inlet velocities, the static pressure does not fall below the vapor pressure of the water. Hence cavitation does not take place in both cases.

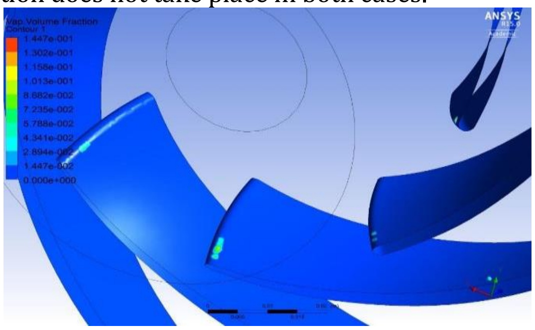

Fig. 5.3 Maximum vapor volume fraction on blades for Q/Q_D =1

Fig. 5.4 shows the maximum vapor volume fraction on blades which is zero. Fig. 5.5 shows the variation of maximum vapor volume fraction versus Q/Q_D ratios. It can be deduced that even at the best efficiency point slight cavitation does occur.

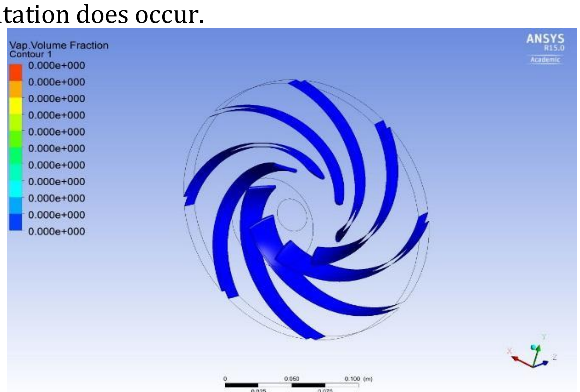

Fig. 5.4 Maximum vapor volume fraction on blades for Q/Q_D = 0.75 and Q/Q_D = 0.48

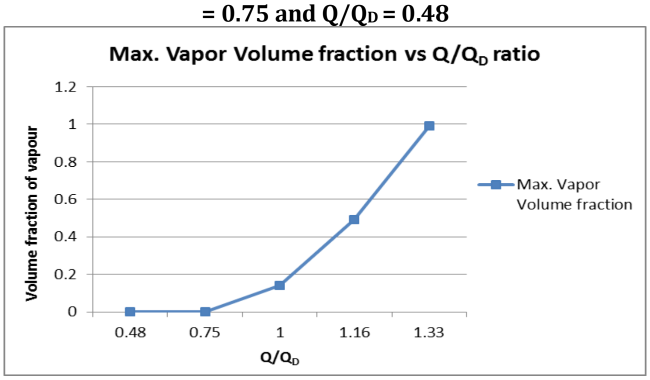

Fig. 5.5 Maximum vapor volume fraction vs Q/Q_D ratio

VI. CONCLUSIONS

The flow behavior of the fluid through centrifugal pump is numerically predicted. Validation of the available experimental results and inspection of the effect of change in variation of parameters along with cavitation phenomenon was numerically studied and simulated. The performance characteristics obtained using simulations show good agreement with available experimental data of the same centrifugal pump with a maximum error of 10%. The vapor bubbles formed near the eye of the impeller show that cavitation inception begins on the blades near the eye where the pressure is low. Numerical results also show that as the velocity near the impeller inlet increases, the static pressure decreases to the point where it falls below vapor pressure and vapor bubbles are formed. Cavitation can be prevented by operating the pump slightly to the left of the best efficiency point.

REFERENCES

[1] J.B. Tonner, J. Tonner, Desalination and energy use, In: C.J. Clevelend, (Ed.),*Encyclopedia of Energy, Vol. 1, Elsevier Inc.*, Amsterdam, 2004. pp. 791–799.

[2] Stepanoff A. J. 1948, Centrifugal and axial flow pumps. John Wiley & son INC.

[3] S. Yokoyama. 1976, "Effect of the position of impeller inlet and lean on the cavitation performance of centrifugal pump," *Fluid Engineering* 12 (10) 601-612.

[4] H. K.Versteeg and W. Malalasekera. 1995, "An introduction to computational fluid dynamics: The finite volume method", John Wiley & sons New York.

[5] LI Xian-hua, ZHANG Shu-jia , ZHU Bao-lin, **HU Qing-bo, 2006, " The Study of the $k-\varepsilon$** Turbulence Model for Numerical Simulation of Centrifugal Pump " *Zhejiang University of Technology.*

[6] ANSYS FLUENT Theory guide Release 15.0, 2013

[7] K.W. Cheah, T. S. Lee, S. H.Winoto, and Z.M. Zhao. 2007, "Research Article Numerical Flow Simulation in a Centrifugal Pump at Design and Off-Design Conditions", Hindawi Publishing Corporation *International Journal of Rotating Machinery* Volume 2007, Article ID 83641, 8 pages

[8] Andreas Peters, Hemant Sagar, Udo Lantermann, Ould el Moctar "Numerical modelling and prediction of cavitation erosion", *Journal of Wear,2015,* pp 189-201.