Scientific Journal of Impact Factor(SJIF): 3.134
e-ISSN(O): 2348-4470
p-ISSN(P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 1,Issue 12, December -2014

# NUMERICAL INVESTIGATION OF CAVITATION OF CENTRIFUGAL PUMP

Eshant S. Rami[1], A.M.Pathan[2]

[1]PG Student (Thermal Engineering), Mechanical Engineering Department, L J Institute of Engineering and Technology, Ahmedabad, Gujarat.,[1]eshantrami@gmail.com

[2]Assi. Professor, Mechanical Engineering Department, L J Institute of Engineering and Technology, Ahmedabad, Gujarat.

**Abstract:** *New computational methods are continuously developed in order to solve problems in different engineering fields. One of these fields is centrifugal pump, where the challenge is to make centrifugal pump more efficient and to reduce cavitation in the pump. One of the main parts of a centrifugal pump that can be improved is the impeller. In order to optimize the centrifugal pump, both experimental and numerical methods are called for. An important topic is here to perform grid sensitivity studies to make sure that the model yields mesh independent results. Another topic of interest is the choice of turbulence model and how this choice affects the grid sensitivity. After this project we made a model that is numerically reliable, mesh independent and fast.*

*This thesis presents a computational study of the flow field generated in centrifugal pump and how that flow field convicts through the impeller. Specifically, the effect that the flow fied acting on the impeller was studied. Data from a modern centrifugal pump manufacturer was used to design a realistic, low speed, large scale efficiency test section. This paper presents the results of computational simulations done in parallel with experimental simulations of the impeller flow field.*

*In comparisons of computational predictions with experimental data, reasonable agreement of the mean flow and generates cavitation in the centrifugal pump. After finding the cavitation area by numerical method, changing the angle of the blade or changing the number of blade the direction of the inlet and outlet flow has been changed.*

**Key words:** *Computational Fluid Dynamics, Centrifugal pump, Impeller, Cavitation number*

## I. INTRODUCTION

Hydrodynamic cavitation describes the process of vaporization, bubble generation and bubble implosion which occurs in a flowing liquid as a result of a decrease and subsequent increase in pressure. Cavitation will only occur if the pressure declines to some point below the saturated vapor pressure of the liquid and subsequent recovery above the vapor pressure. If the recovery pressure is not above the vapor pressure then flashing is said to have occurred. In pipe systems, cavitation typically occurs either as the result of an increase in the kinetic energy (through an area constriction) or an increase in the pipe elevation.

Hydrodynamic cavitation can be produced by passing a liquid through a constricted channel at a specific velocity or by mechanical rotation of an object through a liquid. In the case of the constricted channel and based on the specific (or unique) geometry of the system, the combination of pressure and kinetic energy can create the hydrodynamic cavitation cavern downstream of the local constriction generating high energy cavitation bubbles.

The process of bubble generation, and the subsequent growth and collapse of the cavitation bubbles, results in very high energy densities and in very high temperatures and pressures at the surface of the bubbles for a very short time. The overall liquid medium environment, therefore, remains at ambient conditions. When uncontrolled, cavitation is damaging; by controlling the flow of the cavitation, however, the power can be harnessed and non-destructive. Controlled cavitation can be used to enhance chemical reactions or propagate certain unexpected reactions because free radicals are generated in the process due to disassociation of vapors trapped in the cavitating bubbles.

Orifices and venturi are reported to be widely used for generating cavitation. A venturi has an inherent advantage over an orifice because of its smooth converging and diverging sections, such that that it can generate a higher velocity at the throat for a given pressure drop across it. On the other hand, an orifice has an advantage that it can accommodate more number of holes (larger perimeter of holes) in a given cross sectional area of the pipe.

## II. LITERATURE REVIEW

**Mahesh M.,** *etal* **[1], (2002), "Application of Full Cavitation model to Pumps & Inducers"**

Author represented that a new full cavitation model has been recently developed for performance prediction of co engineering equipment under cavitational flow conditions. The model has been incorporated into an advanced finite volume, pressure-based commercial CFD code that uses hybrid grids to integrate the N-S equations.

The result shows cavitation zones on leading edge-suction side of machine computations on the water jet pump for different non-condensable gas concentration showed sizable changes in the pump developed.

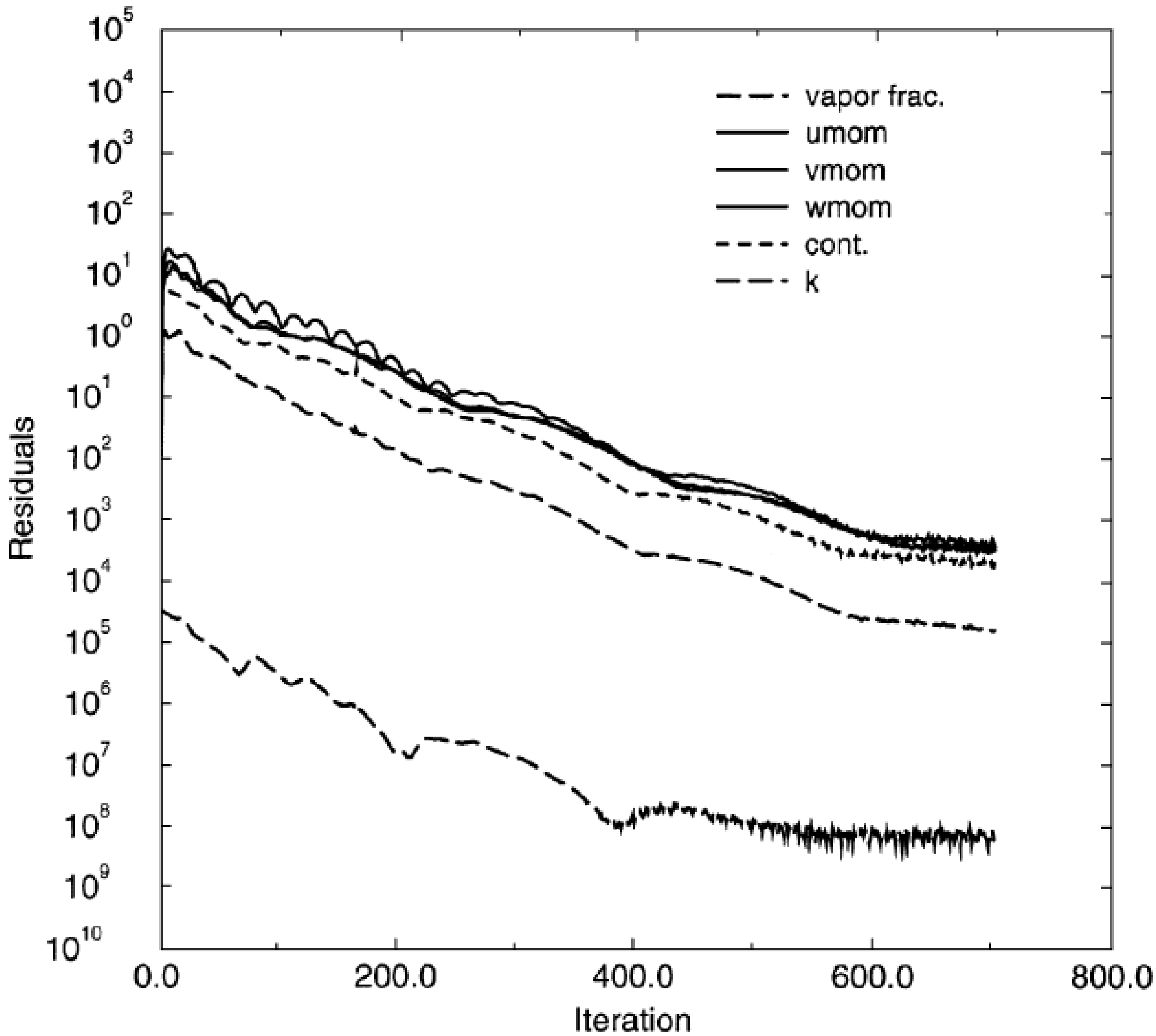



***Fig :- Sample Convergence Plot***

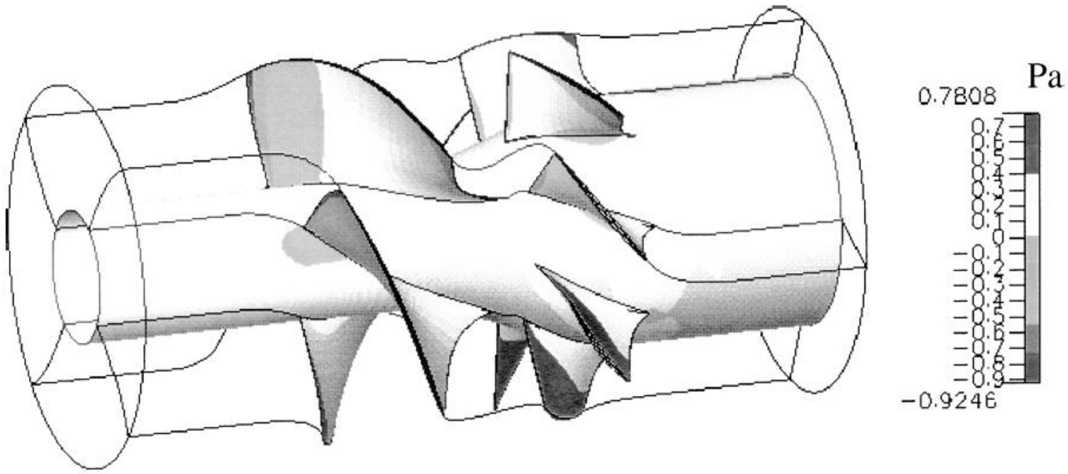
**Dirk Bauer,** *etal* **[2], (2002), "A case study in selective visualization of unsteady 3D flow"**

In this case study they represented that the techniques for the purpose visualizing isolated flow structures in time depending data. The vortex rope can be characterized by high values of normalized helicity, which is a scalar field derived from given CFD data.

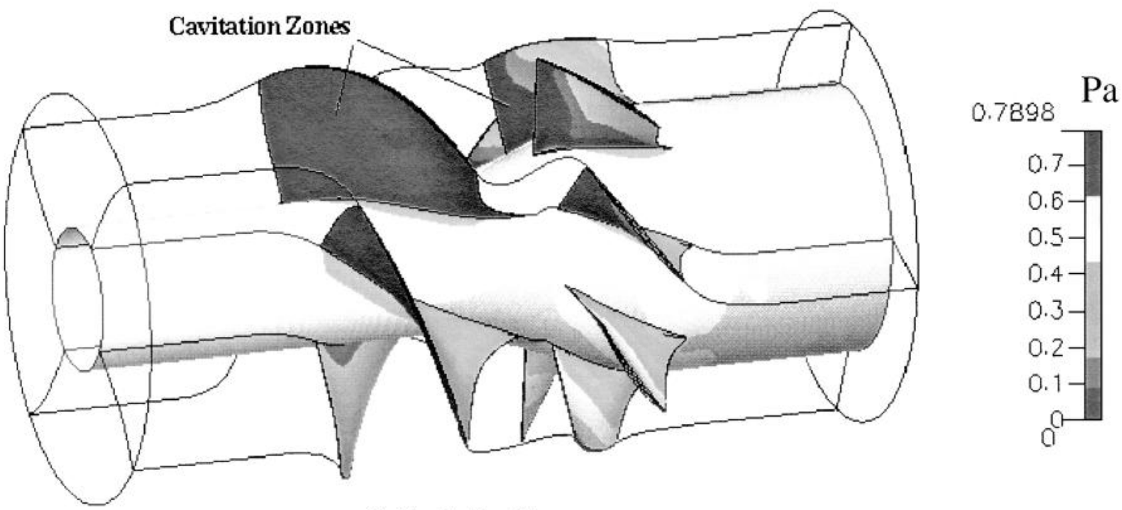
By constraining the visualization to a region of interest occlusion problems are reduced and storage efficiency is gained.

FULL CAVITATION MODEL




A. Single-Phase Solution




B. Cavitating Flow

**Mickel Toussaint, [3], (2004), "Predetermination of Performances of Centrifugal pumps by means of their geometrical characteristics"**

Author represent some application of method, first tom a one stage centrifugal pump fitted with a volute and then to one stage of multicellular water pump.

In this paper the calculation are explained and the expectative performances issued from the calculations compared to experimental results. This can lead to a modification of the design before the concrete relisation of the machine.

**Masamichi Lino,** *etal*, **[4], (2004), "Numerical Analysis of Unstable Phenomena and Stabilizing Modification of an Impeller in Centrifugal Pump"**

They represented the transient 3D internal flow fields in the centrifugal pump and the effect of an impeller have seen analyzed numerically to clarify.

In this paper they compared theoretical head, the simulated head of the pump and impeller show the good stabilizing effect of the modification with particle cutback.

**Jan Matsui,** *etal*, **[5] (2004), "Scale effect to axial thrust of pump turbines and centrifugal pumps"**

Author developed simple software to calculate the pressure distribution and axial thrust of pumps and turbines for very wide range of Reynolds number. The pressure distribution and the thrust of centrifugal pumps are calculated with Reynolds number on them is discussed. The calculation on the water turbine makes clear the dependency of the Reynolds number on axial thrust and leakage flow rate.

**Richard B. Medvitz, [6] (2002), "Performance Analysis of Cavitation Flow in Centrifugal Pump Using Multiphase CFD"**

The author represented that recently developed a multiphase CFD methodology with application focused on sheet and super cavitating flows about under water. As this capability has been mounted and validated for that class of applications. This paper summarized their capabilities and results.

In this paper the theoretical formulation of method is briefly summarized including baseline differential model, specific physical models and numerical methods. The result of Q3D analyses for a 7-blade impeller operating across a range of flow co-efficient and cavitation numbers are presented.

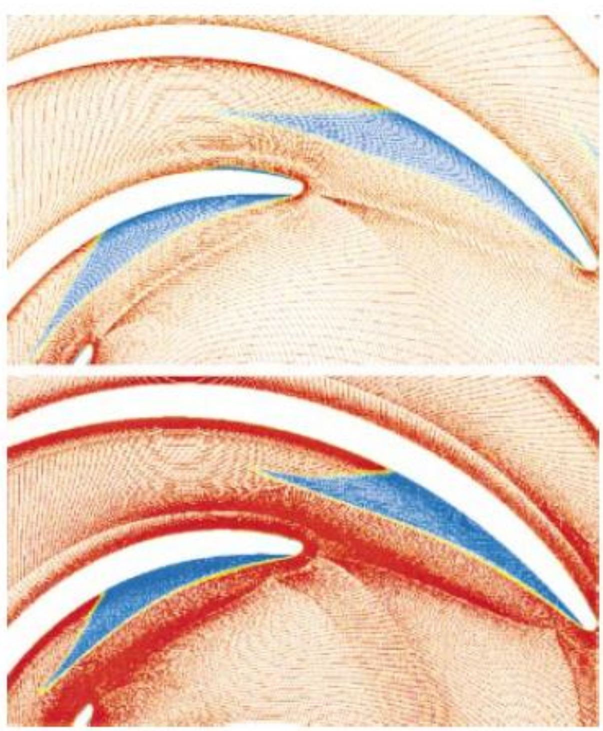


***Fig:- Cavitation bubbles***

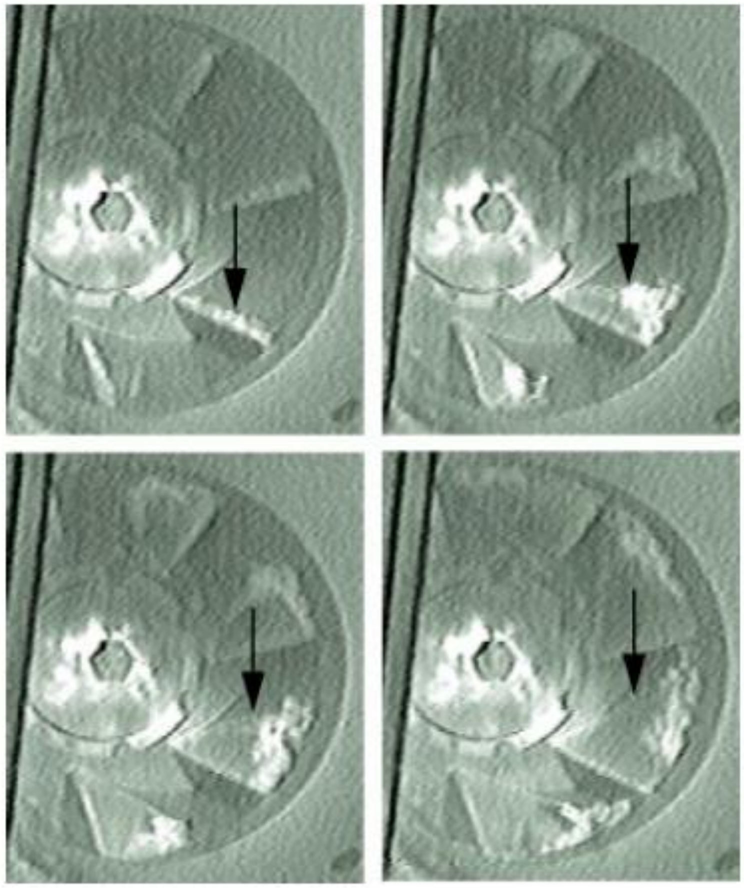


***Fig:- Cavitation Zone***

## III. CONCLUSION

After understanding all this above literatures, we can conclude that cavitation is a serious problem in the centrifugal pump. By using on CFD analysis we can find the area of the cavitation zone. And using different equations in the analysis we can short out how to reduce cavitation in the pump like N-S equation, Reynolds number, 3D model etc.

It was concluded that to improve efficiency, we should look for a way to reduce cavitation in pump. Analys of the cavitational zone is very important respect to the pump. This kind of improvement in pump gives high erefficiency and long life.

## REFERENCES

[1] Mahesh M., H. Y. Li, Ashok Singhal, "Application of Full Cavitation model to Pumps & Inducers", International journal of rotating machinery, 8(1), 2002
[2] Dirk Bouer, Ronald Peikent, Mie Sato, Mirjam Sick, "A case study in selective visualization of unsteady 3D flow", IEEE Visualization, 2002
[3] Mickel Toussaint, "Predetermination of Performances of Centrifugal pumps by means of their geometrical characteristics", 22nd IAHR Symposium on Hydraulic Machinery, 2004
[4] Masamichi Lio, Kazuhiro Tauaka, Takeshi okubo, "Numerical Analysis of Unstable Phenomena and Stabilizing Modification of an Impeller in Centrifugal Pump", 22nd IAHR Symposium on Hydraulic Machinery, 2004
[5] Jun Matsui, Junichi Mokawa, Young-Do Choi, "Scale effect to axial thrust of pump turbines and centrifugal pumps", 22nd IAHR Symposium on Hydraulic Machinery, 2004
[6] Richard B. Meddvitz, Roburt F. Kunz, "Performance Analysis of Cavitation Flow in Centrifugal Pump Using Multiphase CFD", Journal of Fluid Engineering, vol 124/377, 2002