Intl J Engg Sci Adv Research 2015 Sep;1(3):109-114

Numerical Simulation of Cavitations through Orifice and Effect on Coefficient of Discharge by Changing Aspect Ratio

Shashank Deo Nigam
Department of Mechanical Engineering,
Rama University, Kanpur, India
Nigam_shashank24@yahoo.co.in

A. K.Chauhan
Department of Mechanical Engineering,
KNIT, Sultanpur,
India akc.knit@gmail.com

***Abstract*— In many engineering application, cavitation has been a subject of extensive theoretical and experimental research since it has predominantly been perceived as an undesirable phenomenon. Nurick has been established a relation between cavitation number and the coefficient of discharge in the orifice meter. The same work was performed computationally by G.Palau-Salvador at el in Ansys V6.1 and verified the data in the investigation, simulation of cavitation in multiphase flow has been done. In the present investigation geometry & dimension of the orifice were same as taken by Nurick at el [1976]. The simulation was performed by varying the inlet pressure between 2×10^5 to 2×10^8 Pa and constant outlet pressure 95000 Pa. In this report, the dimensions of orifice have been varied in such a way that surface area remain constant. In the present investigation, numerical prediction of cavitation in the series of orifice with different geometry were compared to the experimental measurements to show the possibilities and performance of the new cavitation model in the commercial Computational Fluid Dynamics (CFD) code FLUENT 14.0. Model predictions for the orifice cases accurately capture cavitation inception and its influence on the orifice discharge coefficient. The new cavitation model in FLUENT 14.0 provides very reliable simulation technique for simple geometries when steady flow is assumed. In this research three different rectangular orifice geometries are designed and total work is divided into three cases according to geometry used. Original geometry is used to validate the results with Nurick at el [1976] and another geometry is used to analyze the flow behavior and Discharge coefficient, C_D using same boundary condition as in the original case. This research is about the analysis of Discharge Coefficient (C_D) computationally over a rectangular orifice.**

Keywords*— *computational fluid dynamic technique, flow simulation, steady flow, unsteady flow.

I. INTRODUCTION

In many engineering applications, cavitation has been the subject of extensive theoretical and experimental research since it has predominantly been perceived as an undesirable phenomenon. This is mainly due to the detrimental effects of cavitation such as erosion, noise and vibrations, caused by the growth and collapse of vapour bubbles. The ability to model cavitating flows has drawn strong interest in CFD community. It covers a wide range of applications, such as pumps, hydraulic turbines, inducers and fuel cavitation in as commonly encountered in fuel injection systems. Fluid machinery is a common application where low pressures are routinely generated by the machine action, e.g. on blade surfaces, with a consequent possibility of cavitation.

Existence of cavitation is often undesired, because it can degrade the device performance, produce undesirable noise, lead to physical damage to the device and affect the structural integrity. Details of the existence, extent and effects of cavitation can be of significant help during the design stages of fluid machinery, in order to minimize cavitation or to account for its effects and optimize the design Different aspects of this complex phenomenon have been explored, including, e.g., cavitation bubble collapse and erosion damage, cavitation acoustics, cloud cavitation, and rotating cavitation [1], [2].

Based on the assumption that the flow is inviscid, various numerical methods have been thus far proposed to simulate cavitating fows; the conformal mapping method, the singularity method, and the panel method. The flow around hydrofoil and within a centrifugal impeller could be calculated using these inviscid flow models.

Experimental observations have revealed that the cavitation appearance relates closely to the viscous phenomena of the liquid-phase, such as the boundary layer and the vortex motion. Recently, viscous flow models, which regard the cavitating flow as the bubbly flow containing spherical bubbles, were introduced to provide highly accurate calculations. In the viscous flow models, the Navier-Stokes equation including cavitation bubble is solved in conjunction with Rayleigh's equation governing the change in the bubble radius.

For several years, numerous researchers have been obtaining experimental data about cavitation inception and development for flow elements such as nozzles, orifices, venturis and Schiebe headforms (Nurick, 1976; Abuaf *et al.*, 1981; Meyer *et al.*, 1992; Stutz and Reboud, 1997)

The use of CFD (Computational Fluid Dynamics) in designing engineering devices has increased over the past few years due to the availability of commercial codes featuring state-of-the-art robust models and the ability to run the code on desktop PC’s. Agro-forestry engineering applications of

CFD have also increased in recent years. Palau-Salvador *et al.* (2004) and Wei *et al.* (2006) are excellent examples of this, as both used the commercial code FLUENT V6.1 (FLUENT, 2001) to predict the hydrodynamic behavior inside the labyrinth of an in-line emitter. Other applications in agricultural applications range from food industry processes (Norton and Sun, 2006; Smale *et al.*, 2006) to farm and greenhouse ventilation (Norton *et al.*, 2007).

Evaluating cavitation requires a multiphase flow model. This model may be based on either a separate treatment of the continuum and particulate phase using an Eulerian/Lagrangian approach (Farrel, 2003; Cerutti *et al.*, 2000) or as a homogeneous fluid using an Eulerian/Eulerian approach (Kubota *et al.*, 1992; Singhal *et al.*, 2002; Xing and Frankel, 2002). Many studies have contributed to improving CFD cavitation models to obtain a more realistic approach to simulating vapor formation, which allows cavitation characteristics to be predicted in the areas most affected inside the hydraulic devices. A flashing flow is a non-recoverable cavitating flow.

This phenomenon is very important in flows with strong thermodynamic effects, such as light water nuclear reactors during accidental loss of cooling (Xu *et al.*, 1997). There have been several experimental studies of flashing flows (Abuaf *et al.*, 1981) and prediction modeling studies (Elias and Chambre, 2000; Muñoz-Cobo *et al.*, 2000). Nevertheless, all the studied models presented considerable differences between their predictions and experimental results, such as the study carried out with the commercial code FLUENT 14.0 by Xing (2002), which concluded that the cavitation model in this code underestimated the upstream pressure in a flashing flow.

Cavitation in agricultural applications plays an important role in the efficient performance of valves (Palau-Salvador *et al.*, 2005), venturis (Manzano and Palau-Salvador, 2005), flowmeters (Palau *et al.*, 2004), sprinklers (Pascal *et al.*, 2006) located in pressure irrigation systems and even in xylem vessels of plants when sap travels long distances from root to leaves (Cochard *et al.*, 2007 or Maheraly *et al.*, 2006).

Computational Fluid Dynamics (CFD) analysis is being increasingly applied in the design and simulation flow of sharp edge orifice. Numerical simulation makes it possible to visualised the flow condition inside the sharp edge orifice meter and provide valuable information about the cavitation in orifice or reduce the experiments in the process of orifice design, a deal of labor and facility will be saved, as well as its shortening design cycle. Therefore, great improvement on different orifice meter design must be achieved by CFD analysis of inner flow inside a orifice meter and following application of its result orifice design processes. The objective of the work is to model and numerically investigate the flow field inside the orifice meter using commercial CFD-FLUENT and analyses the performance and cavitation condition at different flow rate.

The objective of this work is to study the characteristics and performance of the new cavitation model of the commercial code FLUENT 14.0 for predicting cavitation in orifice, in steady condition, highlighting their applications in rural environments. The ability of this new model to predict flashing flow is also investigated.

Cavitation Model: General lines, The cavitation model of FLUENT 14.0 is involves two phases and a certain fraction of non-condensable gases, whose mass fraction must be known in advance. This model takes into account the formation and collapse of the bubbles. This new code improves the old cavitation model in Fluent, where, for instance, bubbles were neither created nor destroyed.

Numerical method: The model equation is solved by using the solver FLUENT14.0. In all cases, structured grid generated using generator software Ansys Meshing is used to mesh the domain. The momentum equations are discretized using both first and second order upwind scheme options. The turbulence models used were the standard SST k-ω according to each particular case [See Launder and Spalding (1972) or Veersteg and Malalasekera (1995) for more information on this topic].

When the cavitation model is activated, the fluid is assumed to be a mixture of two species (liquid and vapor). The vapor fraction f affects the fluid density and its governing equation is

$$\frac{\partial}{\partial t}(\rho_m f)+\nabla.(\rho_m v_v f)=\nabla.(\gamma\nabla f)+R_e-R_c \quad (1)$$

ρ_m is the mixture density, f the mass fraction, ρ the velocity vector of the vapor phase and γ the effective exchange coefficient. the source term R_e and R_c represent vapor generation and collapse rates, which can be expressed as a function of the main flow parameter. The expression used in this cavitation model are the function of static pressure and are given by the two equation (Singhal et al., 2002)

$$R_e = C_e \frac{v_{ch}}{\sigma}\rho_l\rho_v\sqrt{\frac{2\,(P_{sat}-P)}{3\rho_l}}(1-f) \quad (2)$$

$$R_c = C_c \frac{v_{ch}}{\sigma}\rho_l\rho_l\sqrt{\frac{2\,(P-P_{sat})}{3\,\rho_l}}\,f \quad (3)$$

Where C_e and C_c are constants, v_{ch} a characteristic velocity, σ the surface tension of the liquid; ρ_l and ρ_v the liquid and vapor density. The Eq. (3.5) is used when P is smaller than P_{sat} and the Eq. (3.6) is used when P is greater the P_{sat}.

The new cavitataion model provides a wider range of options than the old model. The turbulence model can be selected from all the Reynolds Average Navier Strokes possibilities. In the present study, different simulations have been validated, choosing that option that leads to the best agreement with the experimental data extracted from the literature.

Case studied: Several easy geometries were selected to test and validate the new cavitation model in FLUENT 14.0. These geometries included a rectangular orifice with varying geometries in such a way that its surface area remains constant. Only CFD simulations were carried out. The experimental data, geometries and results were obtained from different papers in the bibliography: Nurick (1976) for the rectangular orifices.

Rectangular orifice: As in the case of the rectangular orifice, Nurick's (1976) experimental data were used. The parameters used to validate the model: discharge coefficient (C_d) and cavitation number (Σ). The geometry of the rectangular orifice modeled. The effect of cavitation in rectangular orifices was experimentally investigated by Nurick (1976). Cavitation occurs when the flow passes through a very small orifice, which produces a high differential pressure. This effect can be observed in hydraulic valves (Palau-Salvador *et al.*, 2005) or in flow-meters (Palau *et al.*, 2004).

In Nurick's paper, a large number of experiments were carried out on different geometries, and the experimental results were compared to those obtained by modeling the same geometries. In the present paper, only the geometry shown in Figure 1 is presented, but good agreement was also obtained for the other geometries studied by Nurick (1976).

The turbulence models used were the SST k-ω. It presented good results and no differences were detected between them in the prediction of cavitation in the rectangular orifice. Uniform inlet and outlet static pressure were adopted as boundary conditions. The exit pressure was fixed at 9,5000 Pa and the upstream pressure varied, as in Nurick's experiments, between 2 x 10^8 and 2 x 10^5 Pa. The parameters used to validate the model were the cavitation number (Σ) and the discharge coefficient (C_d):

$$\text{Cavitation number: } \sum = \frac{P_o - P_{sat}}{P_o - P_b}$$

Discharge coefficient:

$$C_d = \frac{\bar{v}_2}{\sqrt{2(P_o - P_b)/\rho}} = \frac{\dot{m}_{actual}}{\dot{m}_{ideal}}$$

where P_o, P_{sat} and P_b are the upstream, vapor and exit pressure, respectively; v_b the velocity at the outlet; ρ the liquid density and m the mass flow.

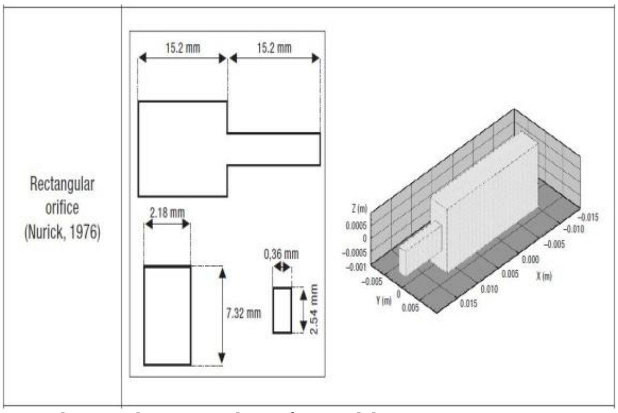


Figure 1 Showing the rectangular orifice model.

The Other CAD model for Orifice geometry was also design in Design Modular by inspiring Nurick's model having constant length of the orifice 15.2 mm but having ratio 1:1which means now the dimensions are (4x4x15.2) mm^3 the total volume of the area is remain constant, 1:3 now the dimensions are (6.93x2.31x15.2) mm^3 having same volume and 1:5 ratio with dimensions (8.944x1.788x15.2) mm^3 height: breath dimension the same we have done in the orifice outlet dimension which is shown in figure 2.

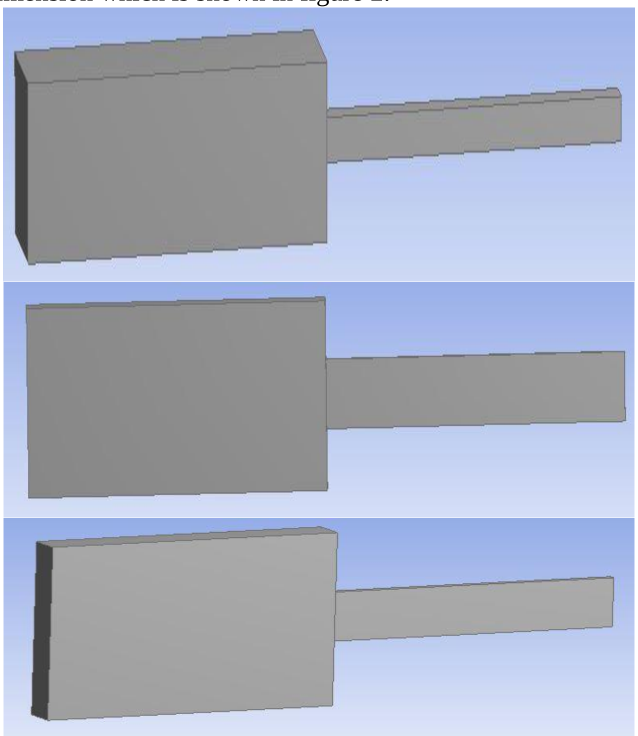


Figure 2 Different CAD model of Orifice geometry.

An unstructured grid with Multi zone cells is used for both the geometry. Optimal number of cells used for Orifice with three different geometries is approx. 5 lakh respectively. The skewness in both the models is coming around 0.39 and 0.42 respectively. Skewness is one of the primary quality measures for a mesh and it determines how close to ideal, a face or cell is. For this study all the simulations were performed in the steady-state mode. The steady-state runs are done for a sufficient number of iterations until the flow data has converged to a constant solution. The convergence criteria were taken as 1×10^{-4} for all the case.

II. Variation in Coefficient of Discharge with Different Geometries:

The simulation results on the Orifice geometry obtained from this investigation. Now the investigation is done on same geometry as of Nurick's Experiment used of 15 mm orifice length and the dimension ratio is 1:1 having same surface area. This data also follow the same pattern but having the different values of coefficient of discharge and this value of coefficient of discharge is not constant, follow the decreasing pattern.

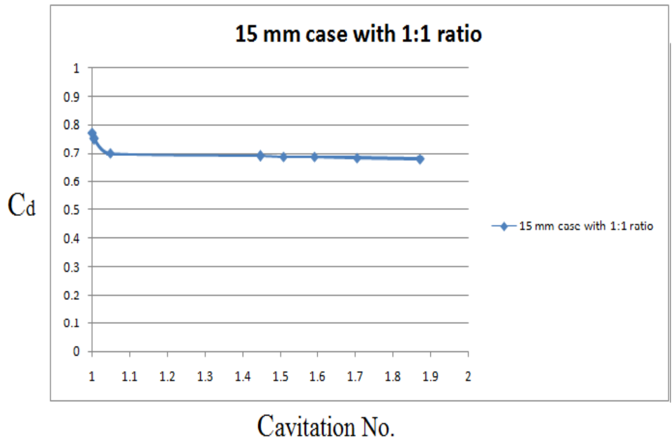


Figure 3 Computational result based on 15 mm length at 1:1 aspect ratio.

Now investigation is done on same geometry of 15 mm orifice length but now the dimension ratio is 1:3 having same surface area. This result show the different value of coefficient of discharge with different inlet pressure having the orifice of 15 mm as taken by Nurick's experiment but dimensional variation is in 1:3 ratios.

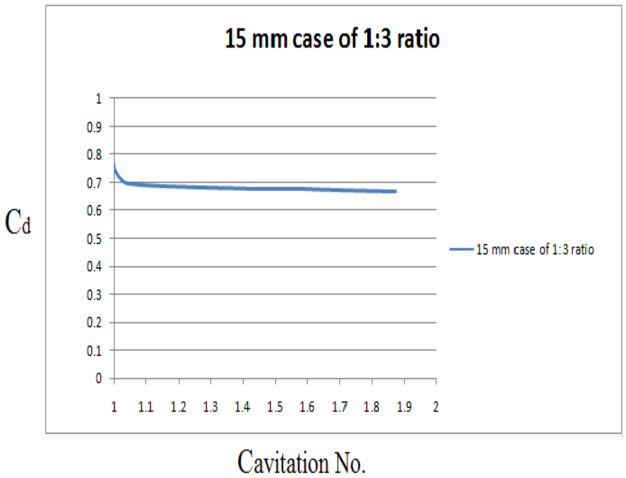


Figure 4 Computational result based on 15 mm length at 1:3 aspect ratio.

Again data is concluded with having the same original geometry of orifice length of 15 mm but now the dimensional ratio are at the ratio of 1:5, and again this data is shown in the Figure 5.6 with the different values of coefficient of discharge. This result show the different value of coefficient of discharge with different inlet pressure having the orifice of 15 mm as taken by Nurick's experiment but dimensional variation is in 1:5 ratios.

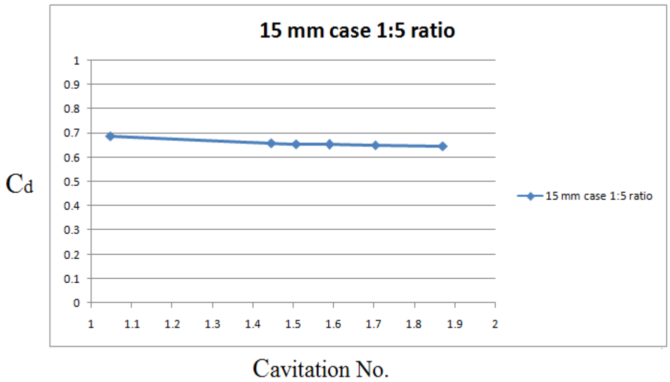


Figure 5 Computational result based on 15 mm length at 1:5 aspect ratio.

This result is from the figure 4.3 it is clear that all the models are following the same pattern as experimental and SST k-ω follows the same trend with min error. The final mesh used featured 4, 98,930 nodes. The coarse grid did not properly simulate the cavitation generated in the orifice outlet, so the finer mesh was needed in accordance with C_d prediction results. With a considerably refiner mesh (4, 98,930 nodes), the model was in reasonable agreement with the experimental data by Nurick (1976).

Now investigating this result with the same orifice length 15 mm but having the different aspect ratio which is 1:1, 1:3 and 1:5.Comparing this result in terms of coefficient of discharge or the variation of discharge with respect to cavitation number.

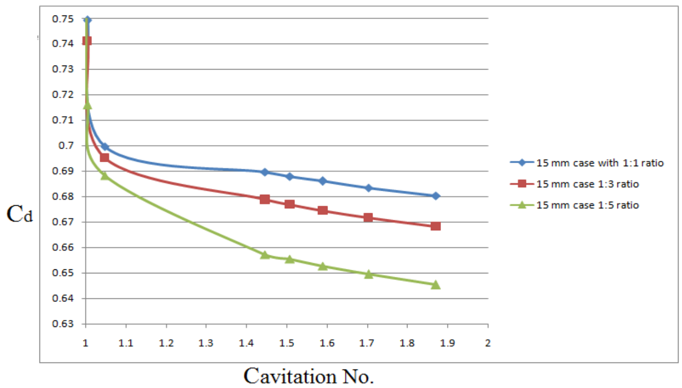


Figure 6 Comparison of computational result with different aspect ratio in 15 mm length.

In all the cases by comparing the result in the same figure having same orifice length with different aspect ratio, the result shows that as aspect ratio increases the coefficient of discharge decreases. The geometry will changes with aspect ratio 1:1 to 1:5 ratios discharge decreases gradually.

Validation of the new geometry: Now the investigation is performed to achieve the better coefficient of discharge to minimize the losses by changing the geometry of the orifice meter. The result shows that as aspect ratio increases the coefficient of discharge decreases.

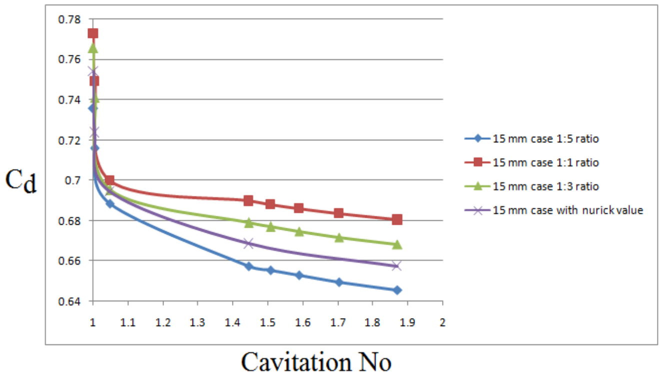
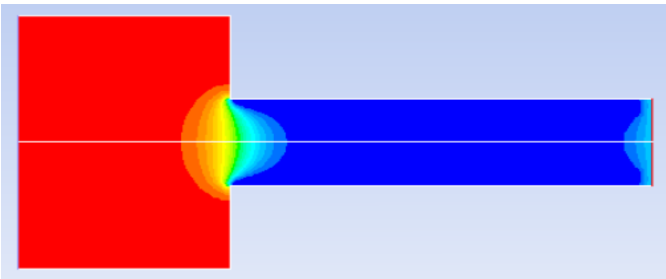


Figure 7 Comparison of computational result of different aspect ratio with nurick's dimensional values.

So by comparing this result which comes out by the new geometry to the Nurick's experimental geometry the conclusion has done that using the equal aspect ratio of the orifice meter, experiment shows the maximum coefficient of discharge and reduces the losses in the orifice meter which increases the optimum result at the outlet of the orifice.

Figure shown the contour of static pressure in the rectangular orifice of the original dimension taken by Nurick in which pressure variation is started to the inlet and followed all over the orifice. The pressure taken at inlet is constant and found the contour of static pressure almost same at different pressure variation in the limit of $2x10^5$ to $2x10^8$ Pa.



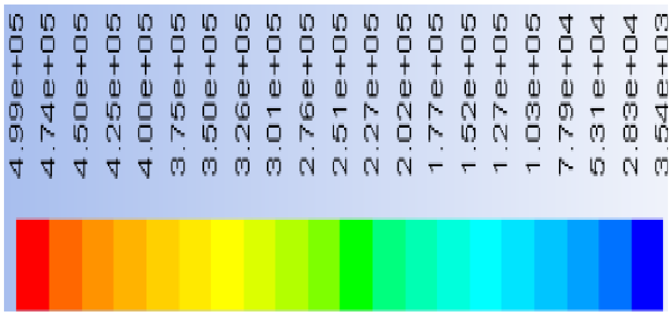


Figure 8 showing Contour of Static pressure

The Extreme or starting pressure is shown by the red zone and as the flow progress the pressure will reduces simultaneously and at the vena contracta the pressure reduces below the atmospheric pressure and create the zone of cavity at the particular region in the orifice and pressure again reduces up to the outlet of the orifice. The static pressure will again increase and reaches the atmospheric pressure when the fluid is just leaving the orifice.

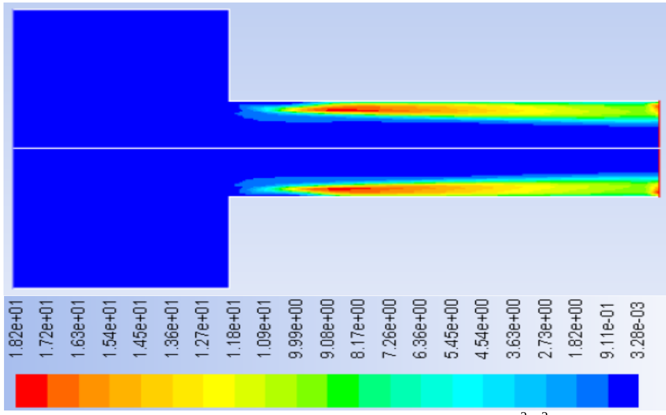


Figure 9 showing Contour of turbulent kinetic energy (k) (m^2/s^2).

In this figure the variation in turbulent kinetic energy inside the orifice has shown and the maximum region inside the orifice is in a blue contour in which turbulent kinetic energy is minimum zone and at the cavity region it become higher and it mainly become greater near the edge of the boundary region or near the surface of the orifice.

The vapor volume fraction is the amount of vapor or cavity formed over total volume contains by the orifice during pressure reduction in the rectangular orifice at vena contracta and this quantity is known as vapor volume fraction. This ANSYS figure shows that the vapor content formed at boundary wall of orifice and this shows the reduction in the pressure at this zone and makes it cavitating.

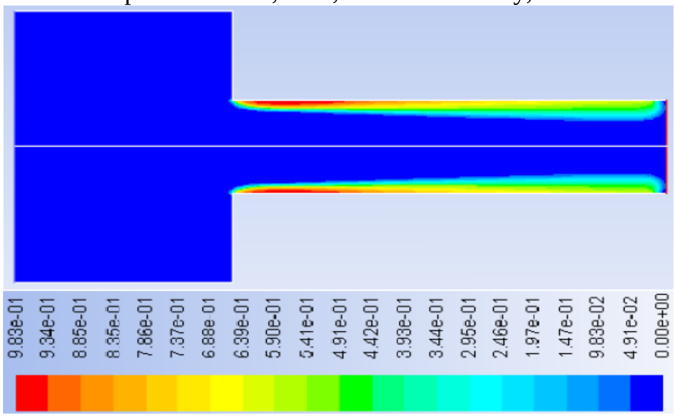


Figure 10 showing Contour of vapor volume fraction.

Percentage loss of energy according to ideal fluid: If we take the initial velocity of the fluid is zero and calculating the final coefficient of discharge. It comes out to be unity according to Bernoulli but the actual coefficient of discharge is almost 0.6 to 0.7 that is because of the total losses occurring to the exit including friction, Borda- Carnot and the turbulence losses. Due to design of different orifice there is different vapor zone or cavity created at the vena contracta and finds the loss in energy with respect to the ideal flow through the Bernoulli theorem. The percentage loss is comes out to be almost 30 to 40% occurred in rectangular orifice used.

So the design preferred by Nurick is appropriate and useful in scientific purpose but the other new different orifice preferred by present investigation minimise the losses and gives the better result in comparison to Nurick geometry. The present investigation can also check different geometries of different dimension by this observation and analysis.

III. CONCLUSIONS

The investigation in the present study leads to the following major conclusions:

- Coefficient of discharge C_D for the given rectangular orifice is coming out be 0.60 to 0.70.
- Coefficient of discharge C_D for the given rectangular orifice can be further increased in between the pressure range of 2×10^6 to 2×10^8 is up to 0.8.
- As the ratio increases from 1:1 to 1:5 the coefficient of discharge decreases in a very narrow range with the same orifice length.
- The significant increment in coefficient of discharge of around 4% is being achieved by using this new geometry with aspect ratio 1:1 at 15mm orifice length.

Following recommendations are listed below based on the computational analysis presented here.

- In the present study ground is considered as rectangular orifice. For further study it can be taken as circular orifice, circular nozzle and also rectangular venturi with the assumption of Steady as well as unsteady flow simulation in the geometry with the same pressure variation. This can help in analyzing the realistic flow visualizing for a running orifice.
- Optimum shape, size and the pressure variation at inlet can be analyzed over the rectangular orifice.
- Other turbulence model and unsteady state simulation can be used for better predicting the behaviour of larger and smaller orifice.

ACKNOWLEDGMENT

The authors would like to thank Professor Anuj Jain and his research group from MNNIT Allahabad for their support during the summer of 2012 and 2013. We acknowledge the reviewers for their comments and suggestions.

REFERENCES

[1.] NURICK W.H., 1976. Orifice cavitation and its effect on spray mixing. J Fluids Eng 98, 681-687. SINGHAL A.K., ATHAVALE M.M., HUIYING L., JIANG, L., 2002.

[2.] Mathematical bases and validation of the full cavitation model. J Fluids Eng 124, 617-624.

[3.] FARREL K.J., 2003. Eulerian/Lagrangian analysis for the prediction of cavitation inception. J Fluids Eng 125(1), 46-52.

[4.] ELIAS E., CHAMBRE P.L., 2000. Bubble transport in flashing flow. Int J Multiphase Flow 26(1), 191-206

[5.] COUTIER-DELGOSHA O., FORTES-PATELLA R., REBOUD J.L., 2003. Evaluation of the turbulence model influence on the numerical simulations of unsteady cavitation. J Fluids Eng 125(1), 38-45.

[6.] STUTZ B., REBOUD J.L., 1997. Experiment on unsteady cavitation. Exp Fluids 22, 191-198.

[7.] STUTZ B., REBOUD J.L., 2000. Measurements within unsteady cavitation. Exp Fluids 29, 545-552.

[8.] XU J.L., CHEN T.K., CHEN X.J., 1997. Critical flow in convergent-divergent nozzles with cavity nucleation model. Exp Therm Fluid Sci 14, 166-173

[9.] VEERSTEG H.K., MALALASEKERA W., 1995. An introduction to computational fluid dynamics. The finite volume method. Ed. Longman, England. 257 pp.

[10.] MEYER R.S., BILLET M.L., HOLL J.M., 1992. Freestreamnuclei and travelling bubble cavitation. J Fluids Eng 114, 672-679.

[11.] FLUENT, 2001. Fluent user's guide. In: FLUENT 14.0 Manual, Vol. 1-21. Fluent Inc., Lebanon.

[12.] Susan-Resiga, R., Bernad, S., Muntean, S., and Anton, I. (2003) Analysis and Development of Cavitating Flow Models and FLUENT Implementation, in Anton, I., Susan-Resiga, R., Sofonea, V., Bernad, S., Muntean S., (eds), Proceedings of the Workshop on Numerical Methods in Fluid Mechanics and FLUENT Applications, Timişoara, pp. 11-21

[13.] Susan-Resiga, R., Muntean, S., Anton I., and Bernad S. (2003), Numerical Investigation of 3D Cavitating Flow in Francis Turbines, Proceedings of the Conference on Modelling Fluid Flow (CMFF'03), Budapest, pp. 950-957.

[14.] Krajnovic, S. Davidson, L., 2004,Large-eddy simulation of the flow around simplified car model. Kuya , Y. , Takeda, K., Zhang , X. , Beeton, S., Pandaleon , T. , 2009, Flow separation control on a race car wing with vortex generators in ground effect Journal of Fluids Engineering, ASME, vol. no. 131 , pp. (121102) 1-8

[15.] Lewis, R., Mosedale, A., Annetts, I., 2009, Using Openfoam and Ansa for road and race car CFD Patrick Gillie´ron , P. , Kourta , A., 2010,