Tikrit Journal of Engineering Sciences/Vol.20/No.2/March 2013, (81-93)

Experimental and Simulation of Dual phase Flow In Venture Convergence-Divergence Nozzle

Dr. Fayadh Mohamed Abed, Professor **Dr. Maki Hag Zedan, Assist. Professor**
Dr.Ghazi-Yousif-Mohammed **Seenaa Khudhayer Salman**
Mechanical Engineering Department -Tikrit University

Abstract

In steam power plants, and high pressure high temperature water flow, phase change takes place resulting in bubbly flow. Such flow causes vibration and noise in the conduits. The present study emphasized on cavitations during a dual phase flow (water-vapor) with a variation of velocities at different conditions in converge-divergence nozzle. The investigation was carried out experimentally and numerically, by CFD simulation. A transparent material is used of PMMA in order to visualize the various regions of the flow. Furthermore, the effect of flow velocities on vibration and noise was evolved in the experimental measurements. The CFD simulation model of this problem is defining a dual compressible viscous flow with *k*-epsilon model for the turbulence modeling. The analyses of the simulation results and the experimental observation have been seen to be comparatively conscionable in the cavitation zone and the estimation of the throat pressure cavitations during a dual phase flow with a variation of mass transfer conditions. A model were combined with a linear viscous turbulent model for the mixed fluids in the computational fluid dynamics software. A CFD Code with modified user intervention is used to simulate steady cavitation. Some of the models were also tested using a three dimensional -CFD code in configurations of cavitation on three-dimensional a converge-diverge sections. The pressure distributions and volume fractions of vapor at different cavitation numbers were simulated, which agreed well with experimental data.

Keywords: Cavitation's, Evaporation, Condensation, and Convergence –divergence Nozzle, bubbly flow.

دراسة تجريبية ومحاكاة عددية لجريان ثنائي الطور عند نفاث ذي مقطع اقتراب–ابتعاد

الخلاصة

في محطات توليد الطاقة البخارية، وارتفاع ضغط المياه ودرجة حرارة تدفق عالية، يحدث تغير الطور مما أدى إلى جريان ذو فقاعات مسببا اهتزاز وضوضاء. في الدراسة الحالية تم التركيز على ظاهرة الفقاعات خلال عملية تغير الطور بين البخار والماء مع تغير السرع عند ظروف مختلفة في نفاث ذي مقطع اقتراب –ابتعاد. الدراسة تمت تجريبيا ونظريا باستخدام الطرق العملية والمحاكاة العددية (CFD). وفي الجانب العملي تم تصنيع نقطع الاختبار من مادة شفافة من PMMA من أجل تصور مختلف مناطق التدفق وملاحظة طول منطقة التحول للجريان ثنائي الطور. علاوة على ذلك ملاحظة تأثير سرعات التدفق على الاهتزاز والضجيج في القياسات التجريبية. وفي نموذج المحاكاة CFD تم ادخال بعض نماذج الجريان ثنائي الطور مع موديل الجريان الاضطرابي. وقد شهدت التحليلات لنتائج المحاكاة والملاحظة التجريبية وجود تطابق نسبيا في منطقة التجويف cavitations وفي تخمين مقدار الضغط في منطقة التجويف الاصغر مساحة في منطقة الخنق خلال المرحلة تدفق الجريان ثنائي

الطور(بخار الماء) مع اختلاف ظروف انتقال الكتلة. تم دخال عدد من الموديلات الخاصة بالجريان ثنائي الطور في الدراسة للمحاكاة العددية لتمثيل ومقارنة الاصلح من النماذج المختارة ليكون الاكثر تطابق مع الجانب العملي والتي تم برمجتها حاسوبيا للتوائم مع نظام المحاكاة وبرنامج الفلونت علما ان الجريان ثلاثي الابعاد ليلائم الشكل المقترح في التصميم. النتائج المستنبطة من المحاكاة والتجريبية من حيث توزيعات الضغط ولنسب الحجمية لكل طور عند قيم مختلفة من مقدار التجويف بينت تطابق جيد.

الكلمات الدالة: والتجويف، التبخر، التكاثف، وفوهة التقارب الاختلاف، جريان الفقاعات

Nomenclatures

Symbols **Unit**

A Face Area [m^2]
A_{cs} Pipe Cross-Sectional Area [m^2]
C_e, C_c **:**Empirical constant 0.02and 0.01
D Tube diameter [m]
f Vapor Mass Fraction
G Acceleration of Gravity [m/s^2]
I Turbulence Intensity [%]
K Turbulent Kinetic Energy [m^2/s^2]
V_{ch} Characteristic Velocity
μ_t Turbulent Viscosity [Ns/m^2]
ρ Density [kg/m^3]
σ Surface Tension [N/m]
ε Dissipation Rate [m^2/s^3]
α Phase Volume Fraction
R_e Evaporation Source Term
R_c Condensation Sink Term
C_{dest} [-] Empirical constant in the condensation term
C_{prod} [-] Empirical constant in the vaporization term
$\dot{m}^+$ *kg/(s.m*^{3}) Vaporization rate
$\dot{m}^-$ *kg/(s.m*^{3}) Condensation rate
$\dot{R}$ *m/s* bubble vapour-liquid interface velocity
R *m* bubble radius
n_b $1/m^3$ number of bubbles per liquid volume
p *Pa* Static pressure
α_v [-] Vapor volume fraction
P_∞ Pa Freestream pressure
P_v Pa Vapor pressure
α_l [-] Lliquid volume fraction
σ [-] Cavitation parameter,
$\sigma = (p_{ref} - p_v)/((1/2)\rho_{ref} V_{ref}^2)$
Vv: m^3 Volume of the vapor phase in a cell
α [-] Vapour volume fraction
γ N/m Surface tension
μ kg.s/m Viscosity
ρ kg/m^3 Mixture density
ρv, ρl kg/m^3 Vapour and liquid densities
ρ_{ref} kg/m^3 Reference density (outlet liquid density)

Subscripts and Superscripts

m mixture
v vapour of vaporization
l liquid
∞ points of large distance from the body
c condensation
e evaporation

Abbreviation

ref reference point
atm. atmospheric conditions
sat. saturation conditions
dest destruction of the phase
prod production of the phase
Mix Mixture

Exp Experimental

Sim Simulation

CFD Computational Fluid dynamic

FVM Finite Volume Method

Introduction

In venture nozzle in which a liquid flow forms gas-filled or vapor-filled cavities under the effect of tensile stress produced by a pressure drop below its vapor pressure is termed cavitation [1]. Cavitation is rife in fluid machinery such as inducers, pumps, turbines, nozzles, marine propellers, hydrofoils, journal bearings, squeeze film dampers etc. due to wide ranging pressure variations along the flow. This phenomenon is largely undesirable due to its negative effects namely noise, vibration, material erosion etc.

The cavitation is departure from usual evaporation, as evaporation, is temperature dependent changing while cavitation is assumed by pressure changing. Cavitation phenomenon can be observed in a wide variety of propulsion and power systems like propellers, pumps, nozzles, valves and injectors. Cavitation is categorized by a dimensionless number called cavitation number, where it depends on the vapor pressure, liquid density, main flow pressure and the main flow velocity. Usually the cavitation formation in a flow is categorized based on the cavitation number of the flow.

The perfect design of the cavitation problems become of interest nowadays. Due to the fast development of computer power during the past decade, a numerical simulation such as CFD has increased enormously, Versteeg[2], a control system and its aspects of cavitation is avoided by CFD which provide a tools that help to know the whole details of cavitation flows criterion.

Numerous modeling strategies have been proposed in the literature, ranging from Rayleigh-Plesset type of bubble formulation, Kubota et al.[3], which separates the liquid and vapor region based on the force balance notion, while the approach of Senocak and Shyy [4], to homogeneous fluid which treats the cavity as a region consisting of continuous composition of liquid and vapor phases.

Further investigations in cavitation, two different approaches have been proposed, an interface tracking model and an interface capturing model. In the interface tracking model, only the equations for the liquid phase are solved, and the vapor phase is not considered. The vapor phase is tracked by the use of interface boundary conditions. The simplest of all cavitation models are broadly classified into potential flow models [5], while The interface capturing approach solves for both phases. The liquid vapor interface is determined using a mixture model, i.e., the cavitating flow is treated as a homogeneous two phase mixture of liquid and its vapor. Most of the interface capturing models are based on a single fluid approach, i.e., the relative motion between the liquid and vapor phases is neglected, and the liquid vapor mixture is treated as a homogeneous medium with variable density. Delannoy and Kueny[6], Song and He and Merkle *et al.*, [7], related mixture density to the local void fraction by a state law. Kubota *et al.*,[3], and Singhal *et al.*, [8] , determined the mixture density by using a supplementary equation relating the void fraction to the dynamic evolution of the bubble cluster. Kunz et al., [9], calculated the mixture density by developing a law for mass transfer between the liquid and the vapor. The advantages of these models are: the model can be applied to most types of cavitation including unsteady cavitation regimes, and a suitable turbulence model can easily be included. The only disadvantage of such models in the determination of an accurate mixture density for the liquid vapor region. The Full Cavitation Model proposed by Singhal *et al*.,[7] is the most popular of all the available cavitation models, and is widely used in industry. Singhal's model is favored for its robustness and generality. These studies can be put mainly into two categories: interface tracking methods[10,11] and homogeneous equilibrium flow models [12–13]. In the first category, the cavity region is generally assumed to have a constant pressure equal to the vapor pressure of the corresponding liquid and the computations are performed only for the liquid phase. In the second category, the single-fluid modeling approach is employed for both phases. Mass and momentum transfers between the two phases are managed either by a Barotropic state law or by a void fraction transport equation**.** Numerical studies and simulations of cavitation have been pursued for years, but it is still a very difficult and challenging task to predict such complex unsteady and two-phase flows with an acceptable accuracy.

Various types of cavitation can be observed based on the flow configurations. In order to predict and control fuel sprays, various theoretical models have been developed. These models need to be validated against available measurements.

The present work: consists of a comparative study between the different models vaporization and condensation approach proposed for the void fraction transport equation in the models of calculations. The models were integrated in a CFD code to represent an overall comparison estimation.

Mathematical Formulations

1. Governing Equations

The flow with possible coexistence of liquid and vapour (and /or gas) is treated as a homogeneous mixture, and the governing equations are the continuity (1) and the momentum (2) equations;

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial(\rho_m u_j)}{\partial x_j} = 0 \qquad \ldots\ldots\ldots\ldots\text{(1)}$$

$$\frac{\partial(\rho_m u_i)}{\partial t} + \frac{\partial(\rho_m u_j u_i)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu_m + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \ldots\ldots\ldots\text{(2)}$$

In multiphase flow the location of any fluid is specified using a volume fraction function (α), and this concept can be written as:

$$\alpha = \begin{cases} 0 & \text{Liquid phase} \\ 0 \langle \alpha \langle 1 & \text{Liquid - Vapor} \\ 1 & \text{Vapor phase} \end{cases}$$

The present paper employs the mixture model, as implemented in the FLUENT commercial code, with the cavitation models that are focused on evaluating the mass transfer as implemented through a User Defined Function wherever applicable in the following section. The mixture density and viscosity are defined as follows based on the vapor volume fraction:

$$\rho_m = \rho_v \alpha_v + (1-\alpha_v)\rho_l$$

$$\mu_m = \mu_v \alpha_v + (1-\alpha_v)\mu_l$$

$$\frac{\partial \alpha_l}{\partial t} + \frac{\partial(\alpha_l u_j)}{\partial x_j} = \dot{m}^+ + \dot{m}^- \quad \text{……..……(3)}$$

2.Cavitation Models

Physically, the cavitation process is governed by thermodynamics and kinetics of the phase change process. The liquid-vapor conversion associated with the cavitation process is modeled through $\dot{m}^+$ and $\dot{m}^-$ terms in Eq. (3), which respectively represent, condensation and evaporation. The particular form of these phase transformation rates, which in case of cryogenic fluids also dictates the heat transfer process which forms the basis of the cavitation model. These modeling are approached as in following paragraph.

2.1 Kunz' Cavitation Model

The mass transfer in Kunz' model [14] is based on two different strategies for creation and destruction of vapor. The evaporation terms are function of the pressure whereas the condensations term are function of the volume fraction [9,14].

$$\dot{m}_e = \frac{C_{\text{Prod}} \rho_v \rho_l}{\frac{1}{2}\rho_l U_m^2 t_\infty} \min(0, p - p_v),$$

$$\dot{m}_c = \frac{C_{dest} \rho_v \alpha_l^2}{t_\infty} (1-\alpha_l) \quad \text{………..(4)}$$

2.2 Singhal Cavitation Model

The model of Singhal termed as the full cavitation model[8]. This model involves two phases and a certain fraction of non-condensable gases, whose mass fraction has to be known beforehand.

$$\left\{ \begin{array}{l} \dot{m}_e = C_{prod} \frac{U_\infty}{\gamma} \rho_l \rho_v \left[-\frac{2}{3} \frac{Min(0, p-p_v)}{\rho_l} \right]^{0.5} (1-f_v) \\ \dot{m}_c = C_{dest} \frac{U_\infty}{\gamma} \rho_l \rho_v \left[\frac{2}{3} \frac{Max(0, p-p_v)}{\rho_l} \right]^{0.5} f_v \end{array} \right\} \quad \text{………………..(5)}$$

2.3 Sauer's Cavitation Model

The main difference between Singhal's and Sauer's model is that Sauer assumes that a constant number of vapor bubbles per unit volume in the liquid flow. The vapor volume fraction α is therefore defined as:

$$\alpha = \frac{n_b \frac{4}{3}\pi R^3}{1 + n_b \frac{4}{3}\pi R^3}$$

As in Singhal model the following simplified version of the Rayleigh-Plesset equation is used

$$\dot{m}_c = \frac{\rho_v \rho_l}{\rho} \frac{3\alpha}{R} (1-\alpha)(p_v - p) \sqrt{\frac{2}{3} \frac{(p_v - p)}{\rho_l}} \quad (p_v - p) > 0$$

$$\dot{m}_e = -\frac{\rho_v \rho_l}{\rho} \frac{3\alpha}{R} (1-\alpha)(p_v - p) \sqrt{\frac{2}{3} \frac{(p_v - p)}{\rho_l}} \quad (p_v - p) < 0 \quad \text{…………………………(6)}$$

2.4 Barotropic Cavitation Model

Due to the local presence of two phases the sonic speed reduces dramatically at the cavitation interface and discontinuities such as shock waves occur in the flow. The barotropic model includes the consequence of these effects. This model does not introduce the vapor volume fraction and hence, the additional equation for mass fraction is not needed. Instead, the density of the fluid is computed from a barotropic state law, When the pressure is higher than the vapor pressure the fluid is supposed to be purely liquid and the density is defined by the Tait equation:

$\frac{\rho}{\rho_l} = \sqrt[n]{\frac{p + p_0}{p_{outlet} + p_0}}$ where $p_0 = 3x10^8$ and n=7 for water. Otherwise the density can then be computed from the ideal gas law, The density of the state between these two limits (mixture of vapor and liquid) is calculated from a smooth curve connecting the two pure phases[15].

$$\rho = \rho_V + (\rho_l - \rho_V) \cdot \frac{1}{\left(1.+Exp\left(-k*\left(\left(p-p_V+1500\right)*14/3000-7\right)\right)\right)} \quad \ldots(7)$$

Numerical Method

The mixture model has enjoyed success with gas-liquid and liquid-granular mixtures of all types. It forms the basis of the cavitations model, which allows for mass transfer due to pressure tension between liquid and gaseous phases. In this study it is aims to predict the cavitations in a dual phase flow involving comprehensive code of calculation, which is defined the flow by a multi task, including, turbulence k-epsilon, cavitations (mixture), using the orthogonally of the meshes, at the surfaces, at the highest gradient of flow, skewness, aspect ratio of mesh gradient defining the turbulence flow and cavitations, which have a Paramounts importance in defining the two phase flow. Validating these criterions via an experimental observation.

The Fluent code is found to be flexible to introduce terms of user intervention, i.e., User Defined Functions (UDFs) and can be programmed, and dynamically linked with the solver itself. UDFs provide access to field variables, material properties, cell geometry data, customization of boundary conditions, in addition to source terms, and variables monitoring during solver running. Post processing will be done in Tecplot and Matlab VR2010a.

1.Turbulence Model Selection

One of the main aspects of cavitation modeling is the interaction of cavitation and turbulence. Understanding of this strongly coupled interaction is necessary to control the periodic unsteady cavitation dynamics. Due to the turbulent nature of cavitation, an appropriate turbulence models should be used in conjunction with the cavitation model. Popular turbulence models of choice have been variations of the k-epsilon model (Kunz et al.,2000)[14]. Coutier Delgosha *et al.*, (2003) [15] suggested a modification for the calculation of eddy viscosity using a density based damping approach. One such density based turbulence viscosity modification is used in this calculations.

$$\mu_t = f(\rho) C_\mu \frac{k^2}{\varepsilon} \quad where$$

$$f(\rho) = \rho_V + \left(\frac{\rho_V - \rho}{\rho_V - \rho_l}\right)^n (\rho_l - \rho_V) \; n >> 1$$

A value of n=10 was suggested by coutier-Delgosh [14].

2. Fluent Simulation

The most suitable codes found to be in assistance to this problem calculations has been selected to be used is the Fluent.

The flow calculations were performed using Fluent 12.1 The flow was assumed to be steady, compressible (including the secondary phase), and isothermal. For the model discretisation, the SIMPLE scheme was employed for pressure-velocity coupling, first-order up winding for the momentum equations, and first-order up winding for other transport equations (e.g. vapor transport and turbulence modeling equations).

For the cavitations cases, the mixture model-based cavitations model in Fluent 12.1 was used, wherein the primary phase was specified as liquid water, and the secondary phase was water vapor. A no-slip assumption was employed to simplify the phase interaction, and the effects of surface tension and non-condensable gas were included. The solver can be based on the finite volume method (FVM), with segregated-implicit-3D-absolute-cell base-superficial model under an operating condition atmosphere pressure, Multiphase flow (mixture, no slip velocity, cavitations) and a Vaporization pressure (2367.8 Pascal); on condensable gas (1.5e-05), this is the mass fraction of non condensable gas dissolved in the working liquid. The standard k-epsilon model used in conjunction with standard wall functions is a suitable choice for this problem. the bubble number density value is 10000, as recommended by Kubota et al.[8] The other parameters are:

$$\sigma_{H2o} = 0.072[N/m], \mu_{H_2O} = 0.001[Ns/m^2]$$

$$\rho_l = 998.95[kg/m^3], \rho_g = 0.554[kg/m^3]$$

$$\mu_l = 0.00108[Ns/m^2], \mu_g = 1.34.10^{-5}[Ns/m^2]$$

3.Numerical Requirements for calculation

A three dimensional meshes are generated using Gambit. A clustering of meshing in the region of cavitation are taken in account. Considering the cavitations caused by a minimum diameter area converge-diverge regions, diameter area converge-diverge regions, the flow is pressure driven, with an inlet pressure 1.2,1.4,1.6 1.8 and 2.0 with an outlet pressure of 0.6 to 0.9 increasing step 0.1 while 2 bar inlet have at an outlet pressure 0.6 to 1.7 bar step 0.3. Geometrical parameters of the model are shown in Figure 1.

Experimental Setup and Measurements

A simple set up used of a clear PMMA venture is used have a geometrical configuration as in Figure 1. The venture is equipment with a pressurizing pump which deliver a pressure of 4.5 bar, with a 25 mm pipe for delivering the pressurized liquid to the intake nozzle having a control valve and un upstream pressure gauge of a scale (0-4 bar) and throttle pressure gauge with average of (-1 to 0.6) bar. However the outlet pressure gauge is a differential gauge at which measure the net pressure difference between the inlet and out let. The mass transfer rate were obtained for a steady state of pressure, and the quantity of liquid a specified period of time being measured for four consequence runs. The measurement of the caviation length being a variation of inlet pressure range of 1.2, 1.4, 1.6 and 1.8 bar and for every pressure point a range of outlet pressure as from 0.1 up to 0.9 bar in a step change of 0.1 bar. However for inlet pressure of 2.0 bar with 0.6 to 1.7 bar step 0.3 being conducted as for this nozzle the vapour liquid phase separation obtained clearly.

Results and Discussion

For the four cavitation models in problem which are used in thus study are Kunz's, Sauer's, Barotropic and Singhal's Models, the treatment of execution of the simulation are conducted through a numerical test before hand with a convergence history for each. This being observed that the required iteration to have a good phase separation i.e vapor-liquid phase as shown in Fig.2.

All models have a common ground that is of Rayleigh-Plesset equation. Each model concerns with a specific conditional assumption. The Barotropic case required a high pressure and higher velocity so that shock wave can accompanied the flow. The phase separation can be only vapour or liquid and the mixture of two phase turns to be problematic with a treatment of discontinuity.

Sauer assume a constant number of vapour bubble per unit volume of the flowing liquid kept constant during the entire computation. For this reason, this model is for a specific and specified cases. Kunz model based on the assumption of the vapour pressure of either above or below vapor pressure of flowing liquid. This require a free flowing liquid over surfaces and may not describe the nozzled flow.

As far as Singhal Model that assume basic mass that being transferred into bubble, one can easily estimates the number of bubble in the either vapour phase and mixture of vapour to liquid.

The simulation for the above four models the models that can describes the flow with cavitation appears to only Singhal model for this pressure range and the nozzle configuration are co-inconcidable with experimental results as shown in tables 1.a,b,c,d and e and comparative a simulated photo in the nozzle at the stated pressure range as shown in figure.3. the cavitation length being estimated from the contour along the nozzle as pressure profile as in figure.4 by using a simulated image that calculated by Fourier transformed from the pixel density unto a colored profile of the flow which is quite reasonably separate the fluid flow into either liquid or vapour and its mixture as in the simulated images Fig.3. These images compared to photographed results which appeared to have a conicidiable similarity as shown in Fig.5, in the same way comparative images obtained four in question model which appears to have no approached results as of the calculated values as in table.2 as far as the CFD method of calculations it is quite adequate to express cavitation but require further effort in models which envisage a thermo dynamical treatments to observe the evaporation with some solid phase evolved due to temperature changes and the results and considering of the vapour back into liquid.

Conclusions

The present work is a contribution to the physical modeling and numerical simulation of cavitating flow. An analytical approach was first performed in order to compare transport equation models proposed the various authors Kunz's, Sauer's, Barotropic and Singhal's. The resemblance between most of the models was observed. A numerical study using a 3D CFD code was also performed, and several models were tested and compared. The results show a pronounce feature of the cativition behavior in term of position and size. These results being represented by Fourier transform of pixels density so that a comparable simulated to the experimental photographs can be visualized, all in all these calculation carried out by CFD Code. This work shows that CFD is a powerful tool in the prediction of the multiphase flow as far as the cavitations but it requires more to be involved in the thermodynamic calculations. Due to the fact of some of the atomized liquid drops being cooled to an ice temperature which causes a solid - liquid phase criterion flow.

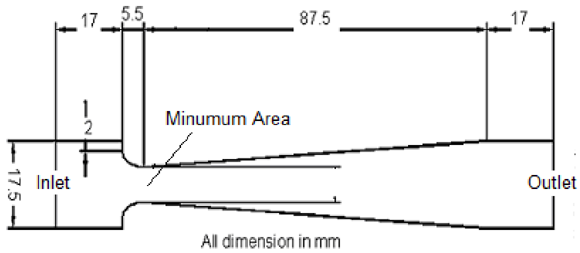
Despite that the phase regions being determined but the re-dissolving back at a later regions is not defined in term of density and temperature. According to this physical interpretation, the CFD calculations can predict a noticeable cavitations.

References

1. Batchelor, G.K., 1967, *An Introduction to Fluid Dynamics*, Cambridge University Press, New York.
2. Versteeg, H. K., Malalasekera W., "An Introduction to Computational Fluid Dynamics – The Finite Volume Method", Longman Group Ltd, 1995
3. Kubota A., Kato H., Yamaguchi H.: *A New Modeling of Cavitating Flows: A Numerical Study of Unsteady Cavitation on a Hydrofoil Section*. J. Fluid Mech. 1992
4. Senocak I., Shyy W.: *Evaluation of cavitation models for Navier-Stokes computations*, Proceeding of FEDSM 02, ASME fluid engineering division summer meeting, Montreal, Canada, 2002
5. Lee, Kinnas and Fine, 1993), and Euler/Navier-Stokes equation-based models (Chen and Heister, 1994; Deshpande *et al.*, 1997.
6. Delannoy and J. L. Kueny, “Cavity flow predictions based on the Euler equations,” in *Proceedings of the ASME Cavitation and Multi-Phase Flow Forum*, vol. 109, pp. 153–158, Portland, Ore, USA, June 1990.
7. C. L. Merkle, J. Z. Feng, and P. E. Buelow, “Computational modeling of the dynamics of sheet cavitation,” in *Proceedings of the 3rd International Symposium on Cavitation (CAV '98)*, pp. 307–311, Grenoble, France, April 1998.
8. A. K. Singhal, M. M. Athavale, H. Li, and Y. Jiang, “Mathematical basis and validation of the full cavitation model,” *Journal of Fluids Engineering*, vol. 124, no. 3, pp. 617–624, 2002.
9. R. F. Kunz, D. A. Boger, D. R. Stineberg, T. S. Chyczewski, J.W. Lindau, and T. R. Govindan, “Multi-phase CFD analysis of natural and ventilated cavitation about submerged bodies,” in *Proceedings of the 3rd ASME/JSME Joint Fluids Engineering Conference (FEDSM ’99)*, p. 1, San Francisco, Calif, USA, July 1999
10. Y. Shen and P. Dimotakis, “The influence of surface cavitation on hydrodynamic forces,” in *Proceedings of the 22nd American Towing Tank Conference (ATTC '89)*, pp. 44–53, St. John's, Canada, August 1989.
11. B. Stutz and J.-L. Reboud, “Two-phase flow structure of sheet cavitation,” *Physics of Fluids*, vol. 9, no. 12, pp. 3678–3686, 1997.
12. J.-B. Leroux, O. Coutier-Delgosha, and J. A. Astolfi, “A joint experimental and numerical study of mechanisms associated to instability of partial cavitation on two-dimensional hydrofoil,” *Physics of Fluids*, vol. 17, no. 5, Article ID 052101, 20 pages, 2005.
13. J.L.Reboud and B. Stutz, “Développement d'un modèle diphasique à deux fluides, analyse expérimentale de poches ventilées,” réf.RT-LT-211708-01 CREMHyG-03.
14. R. F. Kunz, D. A. Boger, D. R. Stinebring, et al., “A preconditioned Navier-Stokes method for two-phase flows with application to cavitation prediction,” *Computers & Fluids*, vol.29, no. 8, pp. 849–875, 2000.

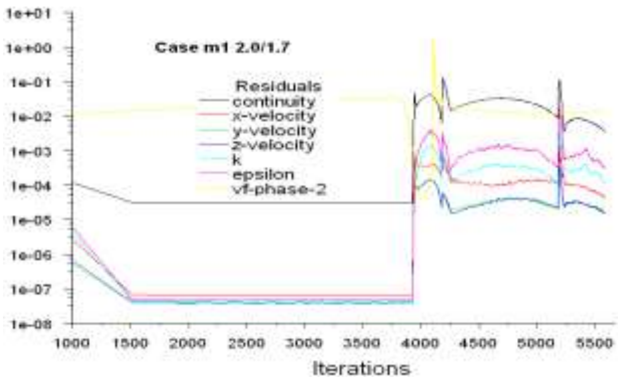
15. Coutier Delgosh, O., Andre Astolfi, J., "Numerical Prediction of the Cavitating Flow on a Two Dimensional Symmetrical Hydrofoil with a Single Fluid Model", Fifth International Symposium on Cavitation, Osaka, Japan (2003),

(a) Photograph

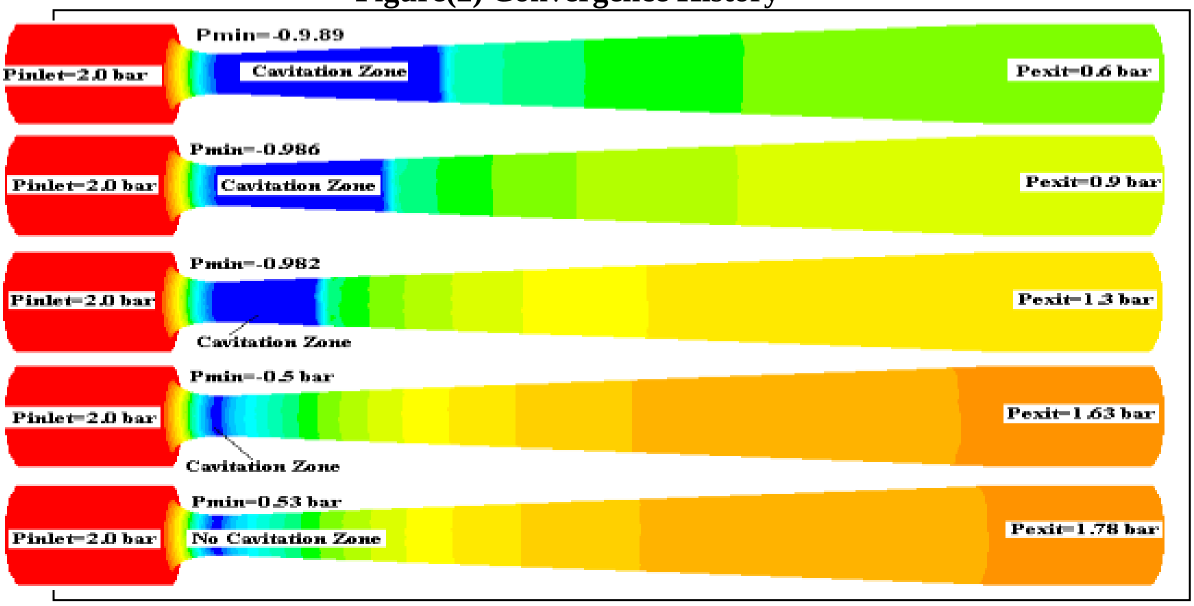


(b) Geometry

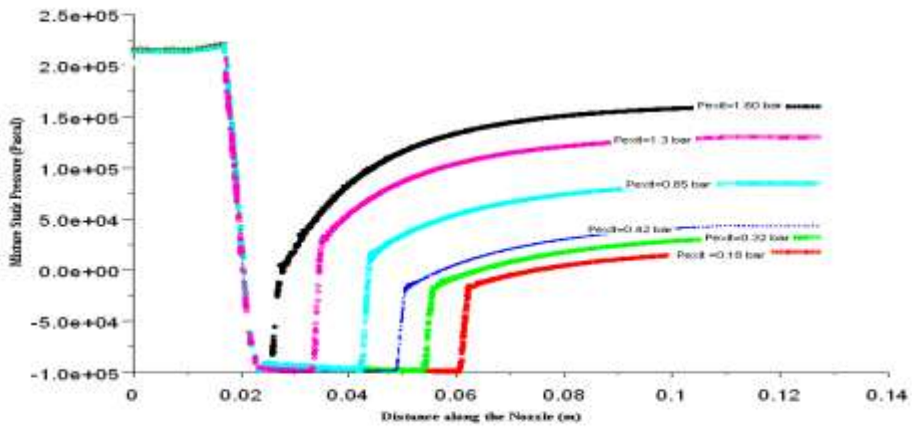
Figure (1) Venture tube Geometry and photograph



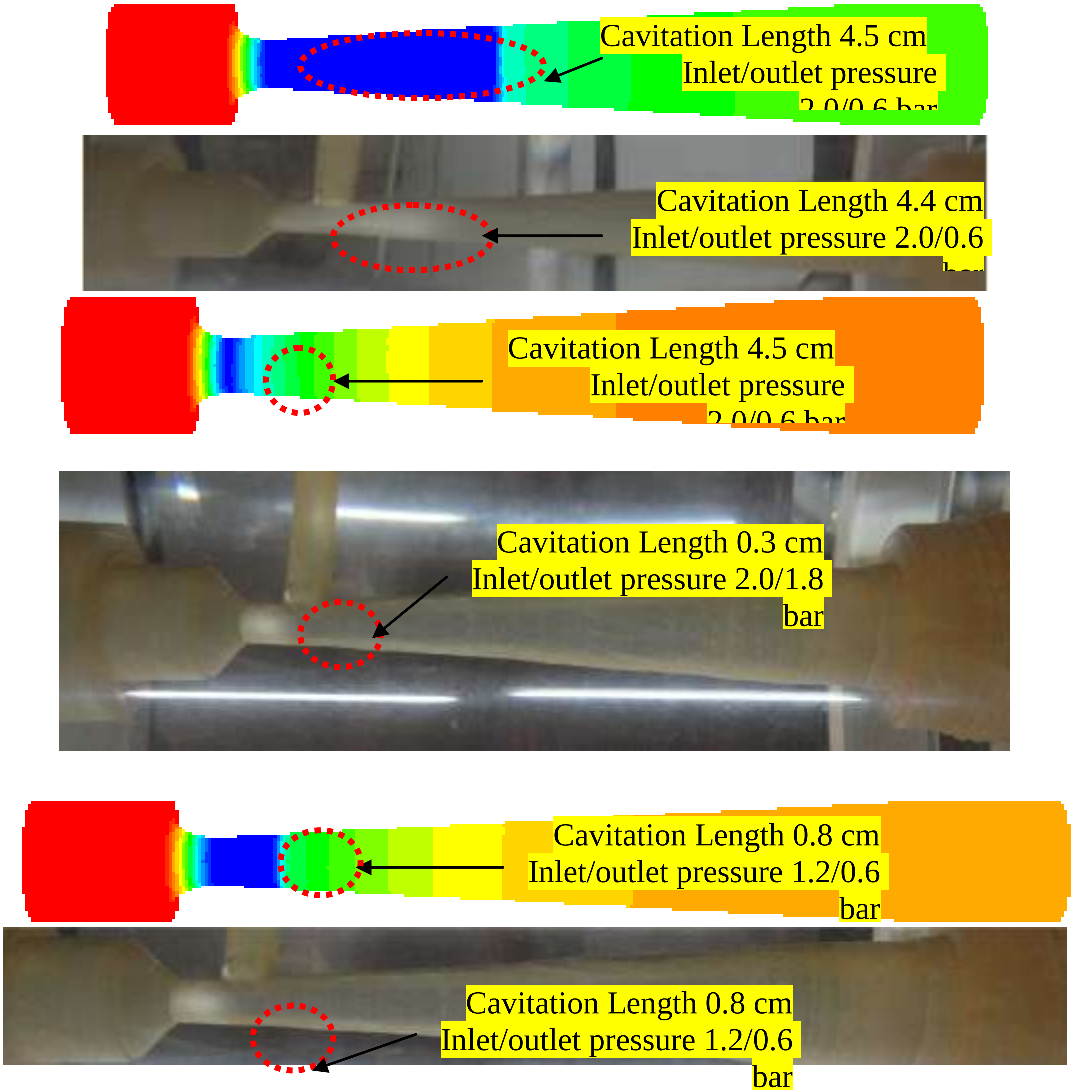
Figure(2) Convergence History



Figure(3) Static Pressure Contours (Code Simulation (table-1a) results)



Figure(4) Pressure Distribution Along the Nozzle(Singhal model)



Figure(5) Comparison of Cavitation model with experimental

Table (1a): Measurement and Numerical Results Comparative

Tests	Test No.1	Test No.2	Test No.3	Test No.4	Test No.5
Inlet/Outlet Pressure(bar)	2.0/1.78	2.0/1.63	2.0/1.30	2.0/0.90	2.0/0.60
Mass Flow Rate kg/sec Exp.	0.75	0.92	0.93	0.93	0.93
Mass Flow Rate kg/sec Sim.	0.60	0.752	0.830	0.841	0.864
Throat Pressure (Exp.) (bar)	0.53	-0.96	-0.96	-0.96	-0.96
Throat Pressure (Sim.) (bar)	0.538	-0.70	-0.982	-0.982	-0.989
Experimental Cavitations Length	0.0	5 mm	18 mm	32 mm	40 mm
Simulation Cavitations Length	0.0	6 mm	20 mm	33 mm	37 mm

Table (1b): Measurement and Numerical Results Comparative

Tests	Test No.1	Test No.2	Test No.3	Test No.4	Test No.5
Inlet/Outlet Pressure(bar)	1.8/0.1	1.8/0.3	1.8/0.5	1.8/0.7	1.8/0.9
Mass Flow Rate kg/sec Exp.	0.95	0.93	0.84	0.63	0.43
Mass Flow Rate kg/sec Sim.	0.94	0.91	0.79	0.58	0.39
Vacume Pressure (Exp.) (bar)	-0.88	-0.85	-0.81	-0.80	-0.74
Throat Pressure (Sim.) (bar)	-0.89	--0.87	-0.83	-0.81	-0.77
Experimental Cavitations Length	55 mm	42 mm	33 mm	23 mm	18 mm
Simulation Cavitations Length	58 mm	48 mm	33 mm	32 mm	23 mm

Table(1c): Measurement and Numerical Results Comparative

Tests	Test No.1	Test No.2	Test No.3	Test No.4	Test No.5
Inlet/Outlet Pressure(bar)	1.6/0.1	1.6/0.3	1.6/0.5	1.6/0.7	1.6/0.9
Mass Flow Rate kg/sec Exp.	1.03	0.92	0.69	0.47	0.42
Mass Flow Rate kg/sec Sim.	1.01	0.83	0.63	0.46	0.40
Vacume Pressure (Exp.) (bar)	-0.87	-0.83	-0.81	-0.81	-0.60
Throat Pressure (Sim.) (bar)	-0.98	--0.96	-0.94	-0.92	-0.78
Experimental Cavitations Length	35 mm	27 mm	20 mm	13 mm	5 mm
Simulation Cavitations Length	50 mm	42 mm	35 mm	27 mm	10 mm

Table (1d): Measurement and Numerical Results Comparative

Tests	Test No.1	Test No.2	Test No.3	Test No.4	Test No.5
Inlet/Outlet Pressure(bar)	1.4/0.1	1.4/0.3	1.4/0.5	1.4/0.7	1.4/0.9
Mass Flow Rate kg/sec Exp.	0.90	0.80	0.67	0.49	0.42
Mass Flow Rate kg/sec Sim.	0.80	0.73	0.60	0.47	0.44
Vacume Pressure (Exp.) (bar)	-0.78	-0.75	-0.65	-0.62	-0.55
Throat Pressure (Sim.) (bar)	-0.83	--0.79	-0.75	-0.70	-0.67
Experimental Cavitations Length	30 mm	23 mm	15 mm	7 mm	3 mm
Simulation Cavitations Length	35 mm	28 mm	18 mm	9 mm	6 mm

Table(1e): Measurement and Numerical Results Comparative

Tests	Test No.1	Test No.2	Test No.3	Test No.4	Test No.5
Inlet/Outlet Pressure(bar)	1.2/0.1	1.2/0.3	1.2/0.5	1.2/0.7	1.2/0.9
Mass Flow Rate kg/sec Exp.	0.59	0.55	0.47	0.43	0.39
Mass Flow Rate kg/sec Sim.	0.58	0.54	0.46	0.42	0.37
Vacuum Pressure (Exp.) (bar)	-0.85	-0.81	-0.80	-0.65	-0.45
Throat Pressure (Sim.) (bar)	-0.95	--0.92	-0.85	-0.65	-0. 45
Experimental Cavitations Length	25 mm	18 mm	10 mm	3 mm	1 mm
Simulation Cavitations Length	27 mm	22 mm	13 mm	6 mm	2 mm

Table (2). Comparison of flow filed of cavitation with Various Model

Model	Singhal	Sauer's	Baratropic	Kunz's
Inlet/Outlet Pressure(bar)	1.8/0.1	1.8/0.1	1.8/0.1	1.8/0.1
Mass Flow Rate kg/sec Exp.	0.95	0.88	0.75	0.82
Mass Flow Rate kg/sec Sim.	0.94	0.94	0.94	0.94
Vacuum Pressure (Exp.) (bar)	-0.88	-0.81	-0.80	-0.65
Throat Pressure (Sim.) (bar)	-0.89	-0.89	-0.89	-0.89
Experimental Cavitations Length	55 mm	50 mm	45 mm	44 mm
Simulation Cavitations Length	58 mm	58 mm	58 mm	58 mm