ASSESSMENT OF TWO PHASE FLOW IN A VENTURE CONVERGENT- DIVERGENT NOZZLE.

Dr. Fayadh M. Abed Al-Dulaimy
Assistant Prof.
Collage of Computer and Mathematics

Dr.Ghazi-Yousif-Mohammed
Lecturer
Mech.Eng. Dept

Tikrit University

ABSTRACT

The present study emphasized on the information of cavitations during the dual phase flow i.e. (water and vapor) in venture converge-diverge nozzle. The choice of nozzle with a transparent material (PMMA), was found suitable for the observation and measurements. The model of this problem of defining dual compressible viscous flow, and k-epsilon model. The comparisons of numerical calculation and experimental observation were found to be comparatively coincidable in cavitational zone and throat pressure, and fractional phase flow.

KEYWORDS

Venture converge diverge nozzle, Turbulence model, Cavitations, Two Phase Flow

NOMENCLATURE

Symbols		Unit
A	Face Area	[m^2]
A_{cs}	Pipe Cross-Sectional Area	[m^2]
C_e, C_c:	Empirical constant	0.02 and 0.01
D	Tube diameter	[mm]
f	Vapor Mass Fraction	
G	Acceleration of Gravity	[m/s^2]
I	Turbulence Intensity	[%]
K	Turbulent Kinetic Energy	[m^2/s^2]
λ	Two-phase flow parameter	[-]
μ	Viscosity	[Ns/m^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]
$\vec{v}_v$	Vapor phase velocity	
γ	Effective Exchange Coefficient	
α	Phase Volume Fraction	

R_e Evaporation Source Term

R_c Condensation Sink Term

ABBREVIATIONS

Mix Mixture

ncg Non-Condensable Gasses

Exp. Experimental

Sim. Simulation

CFD Computational Fluiddynamic

FVM Finite Volume Method

INTRODUCTION

Multiphase flow is all around us. It is cloud formations in the sky, sandy water at the beach, snowstorms, ocean waves, and the morning mist rising from lakes and rivers. As in nature, multiphase flow is common in industrial habitats as well as in oil separators, paper pulp slurries, fluidized coal burners, wastewater treatment plants, fuel injectors, and cavitating in pumps, Singhal [1].

A few attempts have been made to predict cavitating nozzle flow numerically. Delannoy and Kueny[2] used a TVD scheme and a barotropic equation of state to predict inviscid cavitating flow through a venture. Avva[3] presented a model of cavitating flow which was based on an energy equation for a homogenous mixture of phases, assuming no interphase slip.

Cavitation is known to cause unwanted noise and in particular unwanted damage to most hydraulic machinery. It occurs in pumps, propellers, impellers, and in the vascular tissues of plants. Due to the fast development of computer power during the past decade, the usage of CFD has increased enormously, Versteeg [4]. In this study one step has taken further by using numerical tools. Numerical cavitation modeling is very complex, and a lot of work was put down to validate the CFD code, Berntsen [5].

1.The Mixture Model

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, skew ness, aspect ratio of mesh gradient defining the turbulence flow and cavitations, which have a Para mounts importance in defining the two phase flow. Validating these criterions via an experimental observation.

FLUENT SIMULATION

The most suitable codes found to be in assistance to this problem calculations has been selected to be used is the Fluent. Which is widely used in a two phase flow and can be defining the turbulence and cavitation models with a suitable approaches.

The flow calculations were performed using Fluent 6.1[9] 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 6.1[9] 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. Additional information on the cavitations model is provided in Ref.[1], to carry out the numerical calculations necessary to produce satisfactory simulations of the flow problem. The solver can be based on the finite volume method (FVM), with segregated-implicit-3D-absolute-cell base-superficial model under an operating condition 101000 Pascal and Gravity is -9.81 m/s^2 , 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 vaporization pressure is a property of the working liquid, which depends mainly on the liquid temperature. The default value is the vaporization pressure of water at a temperature of 300 K. 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. [6] The other parameters are:

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

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

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

Another requirements evolved in this code is a certain relaxation factors has to be modified to satisfy the stability of the calculation as shown in table 1.

1.Problem Description and Mesh Generation

The simulations were caried out with Fluent 6.1[9] with 3-dimensional meshes, which were generated using Gambit. The mesh is (clustering) in the region of cavitation (liquid was vapor) and coarse in the region of water flow.

Considering the cavitations caused by a minimum diameter area converge-diverge regions, the flow is pressure driven, with an inlet pressure with various values 2.0 and 2.2 bar and an outlet pressure of values (0.6, 0.9, 1.3, 1.63 and 1.78) and (0.18, 0.32, 0.42, 0.85, 1.3, and 1.60) respectively. Geometrical parameters of the model are shown in Figure 1. All the boundary conditions are defined as following.

At the inlet, the static pressure was specified, the turbulence intensity was set to 5%, and the volume fraction of vapor was assumed to be zero (for the cavitating cases). For the cavitating cases, the exit absolute pressure was set to prescribed values in order to control the evolution of the cavitations process. At solid walls, the no-slip boundary condition was imposed. The consistency of the number of grid points are tested by examined of the skewness and the aspect ratio of the meshes, based on the requirements of CFD calculations.

2.Approaches of Multiphase Modeling

Based on literatures, there are two approaches for the numerical calculation of multiphase flows: the Euler-Langrange approach and the Euler-Euler approach Yongliang Chen and Stephen D. Heister [7,8]. the most common approaches of multiphase modeling are Euler-Lagrange, Euler-Euler Approach, VOF Model, Eulerian Model and Multiphase mixture model.

2.3 Multiphase Mixture Model.

In the present work, the Mixture Model is most suitable to our calculation. The following items represent the main equations of this model.

1. Continuity equation for the mixture, Avva[3]. At any point in space that a mixture fluid can be defined by weighting the properties and field quantities of phases with the local volume fractions values.

2. Phase Volume Fraction transport equation for all the secondary phases[9]. The volume fraction field for the primary phase is automatically computed from summing up to 1.0 the volume fractions for all phases.

3. Momentum equations for the mixture, Fluent 6.1 [9].

4. Slip velocity equation. The area of application for the Mixture Model is restricted to multiphase flows in which the secondary phase structures reach equilibrium with the surrounding primary phase. A simple algebraic relation between the slip (relative) velocity for each secondary phase is used to define different velocity fields for the phases.

5. Turbulence model equations for the mixture, Fluent 6.1 [9]. There are three k-ε turbulence models implemented with Multiphase models: Standard k-ε, Renormalization Group (RNG) k-ε, and Realizable k-ε. in this study standard is used.

4. Cavitations Modeling With Fluent

Fluent relies on Mixture model with the following assumptions

- Two phases only are involved a liquid (water) and vapor
- Mass fraction of non-condensable gasses.
- Slip velocity is considered.
- The system is isothermal

The continuity equation for the mixture is

$$\frac{\partial}{\partial t}(\rho_m) + \nabla(\rho_m \bar{v}_m) = 0 \quad \text{..................} \quad (1)$$

The mixture obeys a momentum equation given as:

$$\frac{\partial}{\partial t}(\rho_m \bar{v}_m) + \nabla(\rho_m \bar{v}_m \bar{v}_m) = -\nabla p + \nabla \cdot \left[\mu_m \left(\nabla \bar{v}_m + \bar{v}_m^T\right)\right] + \rho_m \bar{g} + \bar{F} + \nabla\left(\sum_{k=1}^{n} \alpha_k \rho_k \bar{v}_{dr.k} \bar{v}_{dr.k}\right) \quad \text{.........} \quad (2)$$

Where n is the number of phases which equals 2.0 and ρ_m is the mixture density:

$$\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k \,, \qquad \mu_m = \sum_{k=1}^{n} \alpha_k \mu_k \,,$$

$$\bar{v}_m = \frac{\sum_{k=1}^{n} \alpha_k \rho_k \bar{v}_k}{\rho_m}$$,and $\bar{v}_{dk}$ are the driftvelocities: $\bar{v}_{dk} = \bar{v}_k - \bar{v}_m$ vapor mass fraction governed by transportequation.

$$\frac{\partial}{\partial t}(\rho f) + \nabla(\rho \vec{v}_v f) = \nabla(\gamma \quad \nabla f) + R_e + R_c \,,$$

$$f = \frac{\alpha_v \rho_v}{\rho} \quad \text{..............................} \quad (3)$$

Rate terms:

$$R_c = C_c \frac{V_{ch}}{\sigma} \rho_l \rho_v \sqrt{\frac{2(p_{sat} - p)}{3\rho_l}} (1 - f), \qquad p \langle p_{sat}$$

$$R_c = C_c \frac{V_{ch}}{\sigma} \rho_l \rho_v \sqrt{\frac{2(p - p_{sat})}{3\rho_l}} f, \qquad p \langle p_{sat}$$

$$where \quad V_{ch} = \sqrt{k}$$

Turbulent Fluctuations

Turbulent pressure fluctuations influence cavitations process modified vapor pressure

$$p_v = 0.5(p_{sat} + p_{turb}) = 0.5(p_{sat} + 0.39\rho k)$$

And this gives new rate expressions

$$R_e = C_c \frac{\sqrt{k}}{\sigma} \rho_l \rho_v \sqrt{\frac{2(p_{sat} - p)}{3\rho_l}} (1 - f_v - f_g), \quad p \langle p_v$$

$$R_c = C_c \frac{\sqrt{k}}{\sigma} \rho_l \rho_v \sqrt{\frac{2(p - p_{sat})}{3\rho_l}} f_v, \quad p \langle p_v$$

EXPERIMENTAL SETUP

The setup of the rig is shown in Figure 1. The experimental data are obtained by setting the inlet and outlet pressure by using two gage pressures the inlet gage fixed upstream the test section and connect with the upstream pipe through visible plastic tube, Gauge pressure at inlet a range of 4 bar maximum.

The outlet gage pressure is a differential gage measure the difference of upstream and downstream pressure with a range of (0 to 4 bar). The pressure at throat section is carried out by using a pressure gage connect directly to the throat area of gage pressure at throat with a range of (–1 to 0.6 bar).

Mass flow rate is measured after specifying the inlet and outlet pressure values and specifying the quantity of the outlet flow water and determined the time of this quantity which is flowing in the test section, and then the value of the mass flow rate can be defined. The measured quantities given in table -2.

Five consequently runs has been performed in order to evaluate the region length at the given run pressure, at which, phase changes can take place in a region of cavitation. For which many observation have to be taking care of such as: At any given time, the liquid quantity has to be measured at the specific pressure which is adjusted accordingly at both two ends on the up and down streams of the venture. The tube is specially made from PMMA which is clear to monitor the flow and phase changes that taken place at the specific noted flow and pressure and can be photograph and photoanalyzed as well as temperature changes at the specific zones due to phase changes by a low temperature sapphire pyrometric recording. An image matching for each run with the calculated case using the Fluent Code which involve both hydrodynamic and thermal library subroutine at high sophistications in terms of numerical and mesh generation analysis with a high flexibility. Thus with this case which involve a phase change a certain equation of state introduced to express the phase changes from liquid to mixed separated- gas -liquid- phases as well as a certain run ice can be noted. These macro routine where introduced to express exactly the physical phenomena that can describe such a flow. A computer type Pentium IV with 3 Giga Hz processor enabled us to evaluate and optimized the matched cases of flow at every given run.

COMPARATIVE RESULTS

The observed experimental photograph shows directly at the stated conditions of the venture nozzle at the same static pressure. this have been evaluated by the calculation of the fluent code, gives the same structure and length of the cavitations evolved due to the degassing of the dissolve gasses in the pressurized liquid. Causing shock-wave and atomizing the liquid having a two phase flow of atomized liquid particles with another phase of liquid. And this defined the cavitations length which appears from the figures of pressure contours and profile to be very near to what the experimental length as shown in Figures 2, 3 and table 1, 2.

The effect of the back pressure at the throat pressure have to have reduction in the cavitations length, with almost the diminished at a final pressure difference of 0.38 bar. This negative pressure is likely to be related to the degassing criterion of the liquid gas system, Brent A. Cullimore, David A. Johnson [10,11].

In term of calculations the stability of calculation is achieved reasonably well as the shock wave occurs immediately as far as the back pressure low, progressively reduced as the back pressure increased. This can be seen in the number of iteration related to the minimized error table1. This is clearly observed as shown in the rapid convergence in case study of Figures 4 and 5. more effectively is the optimization of choice and selection, the grid criterion related to the phase changes, and the geometry profile, further more the quality of the results much dependent on the skew ness and the orthogonally of the grid.

CONCLUSIVE REMARKS

This work show 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 given a different density profile as shown in Figures .2 and 3. Also the evolved gas has to be dissolved back in the liquid given further regions.

According to this physical interpretation, the CFD calculations can predict a noticeable cavitations.

REFERENCES

[1] Singhal, A. K., Athavale H. Y. Li, M., and Jiang. Y."Mathematical Basis and Validation of the Full Cavitation Model." ASME FEDSM'01, New Orleans, Louisiana, 2001.

[2]Y. Delannoy,D and Kueny J. L., "Two Phase Flow Approach in Unsteady Cavitation Modeling," Cavitation and Multiphase Flow Forum, ASME FED, vol. 98, pp. 153-158 1990

[3]. Ram K., Singhal, S. and Dennis H. "An Enthalpy Based Model of Cavitation." ASME FED v. 226, pp. 63-70, 1995.

[4] Versteeg, H. K., Malalasekera W., "An Introduction to Computational Fluid Dynamics – The Finite Volume Method", Longman Group Ltd, 1995

[5] Bernsten, G. S., Kjeldsen, M. & Arndt, R. e. a. "Numerical modeling of sheet and tip vortex cavitation with Fluent 5". Presented at CAV 2001: Fourth International Symposium on Cavitation, California Institute of Technology, Pasadena, CA USA. June 20-23, 2000

[6]. Kubota,A., Kato,H. and Yamaguchi,H. "Finite Difference Analysis of Unsteady Cavitation on a Two-Dimensional Hydrofoil," Proc. of the 5th Int. Conf. on Numerical Ship Hydrodynamics, Hiroshima, pp. 667-683, Sept. 1989.

[7] Yongliang and Stephen D., "A Numerical Treatment for Attached Cavitation," J. of Fluids Engineering, vol. 116, n. 3, pp. 613-618, 1994.

[8] Yongliang and Stephen D., "Two-Phase Modeling of Cavitated Flows," Computers and Fluids v. 24, n. 7, pp. 799- 809, 1995

[9]. Fleunt V6 - User's Guide, Fluent, Inc; 2001.

[10]http//en.wikipidia.org/wiki/cavitation .

[11 Brent A., David A.,” Thermo]hydraulic Solutions for Thermal Control, Propulsion, Fire Suppression, and Environmental Control Systems”, ,C&R Technologies, Inc. Littleton Colorado, USA Paper Number 1999-01-2159

Table-1-Numerical Test (Under relaxation factors)

Properties Under Relaxation	Inlet/Outlet Pressure 2.0/0.6	Inlet/Outlet Pressure. 2.0/0.9	Inlet/Outlet Pressure. 2.0/1.3	Inlet/Outlet Pressure. 2.0/1.63	Inlet/Outlet Pressure. 2.0/1.78
Pressure	0.3/standard	0.3/standard	0.3/standard	0.3/standard	0.3/standard
Momentum	0.3	0.3	0.3	0.1-0.3	0.1-0.3
Vaporization	0.1	0.1	0.1	0.1	0.05-0.08
Volume Fraction	0.1	0.1	0.1	0.05-0.1	0.05-0.1
TKE	0.3-0.5	0.3-0.5	0.3-0.5	0.3-0.5	0.3-0.5
TDE	0.5	0.5	0.5	0.5	0.5
Viscosity	0.5	0.5	0.5	0.5	0.5
Density	1	1	1	1	1
Body	1	1	1	1	1
No. of Iteration	650	700	1350	2000	4500

Table 2a: Measurement and Numerical Results Comparative

Measuring Points	M1	M2	M3	M4	M5
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
Number of Iteration	4500	2000	1350	900	650

Table 2b: Measurement and Numerical Results Comparative

Measuring points	Mass (kg)	Time (sec)	Mass Flow Rate(kg/sec) Exp./Sim	Inlet Pressur (bar)	Throat pressure (bar)Exp	Throat Pressure (bar)Sim	Outlet Pressure bar
M1	60	62.2	0.9646/0.845	2.2	-1	-0.974	1.60
M2	60	62.2	0.9646/0.843	2.2	-1	-0.976	1.3
M3	60	62.2	0.9646/0.885	2.2	-1	-0.985	0.85
M4	60	62.2	0.9646/0.979	2.2	-1	-0.989	0.42
M5	50	53.7	0.9311/0.963	2.2	-0.89	-0.99	0.32
M6	40	59.5	0.6723/0.845	2.2	-0.9	-0.997	0.18

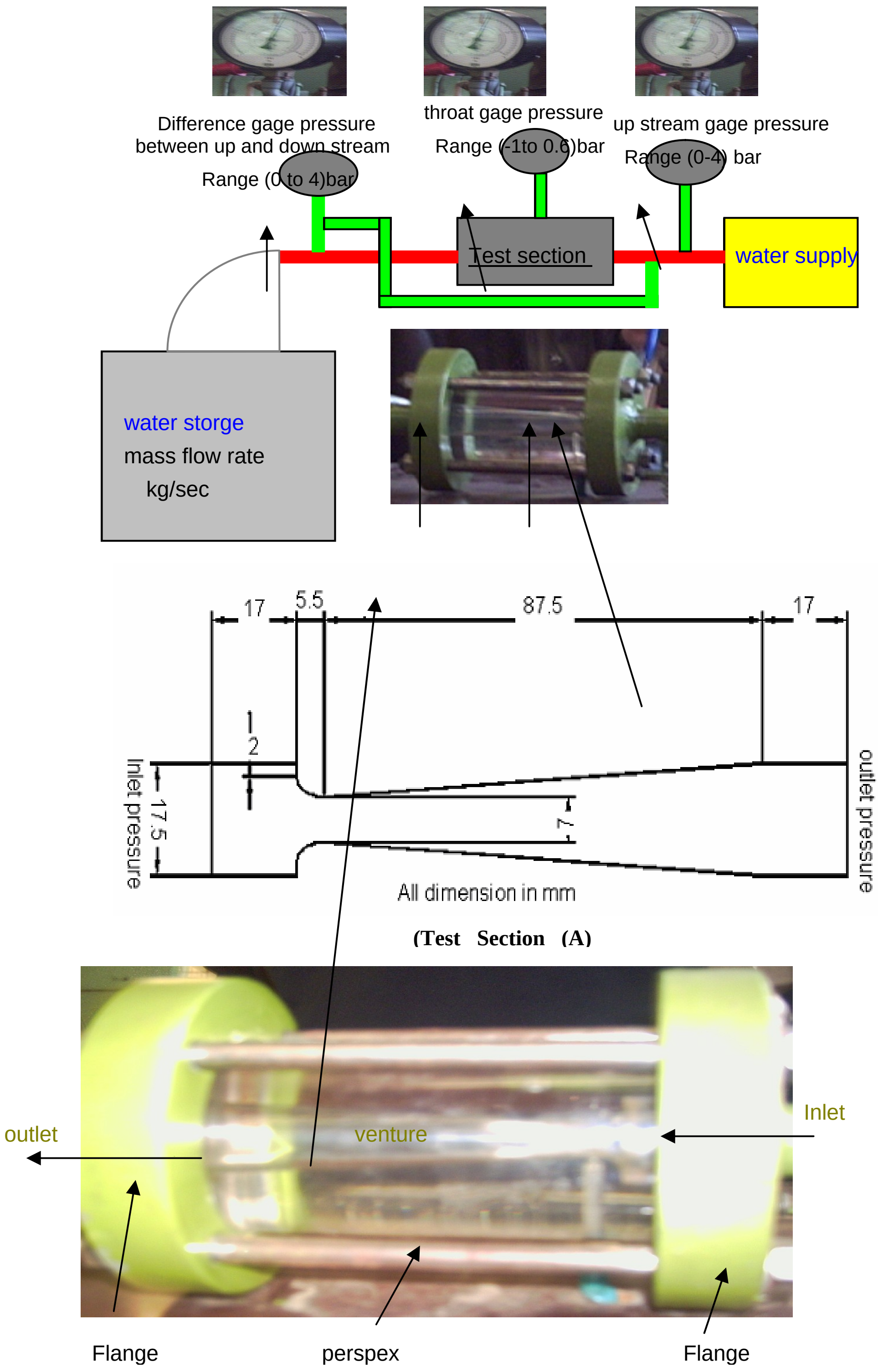

Fig.1 Schematic description of the experimental facility (With photograph that illustrated the Venture part)

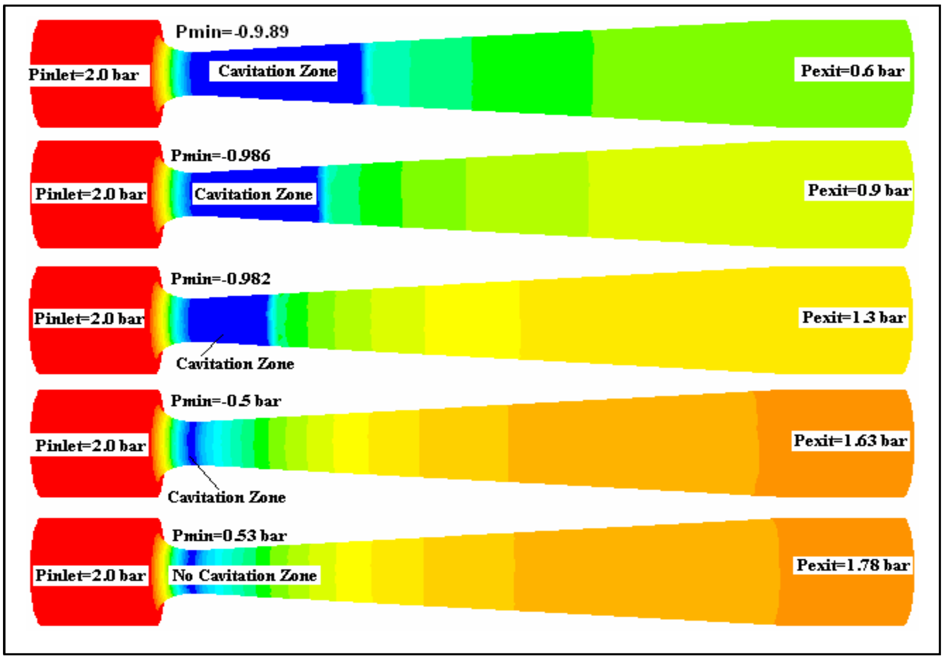

Fig.2a Static Pressure Contours (Code Simulation (table-2a) results)

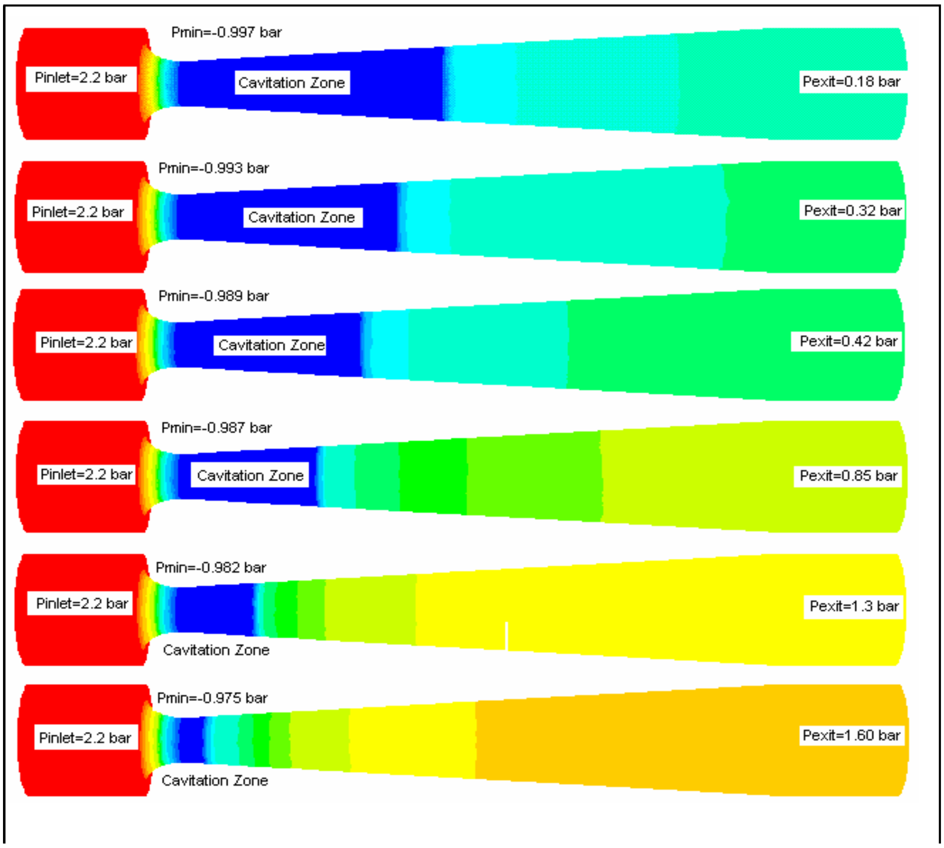

Fig.2b. Static Pressure Contours (Code Simulation-(table-2b) results)

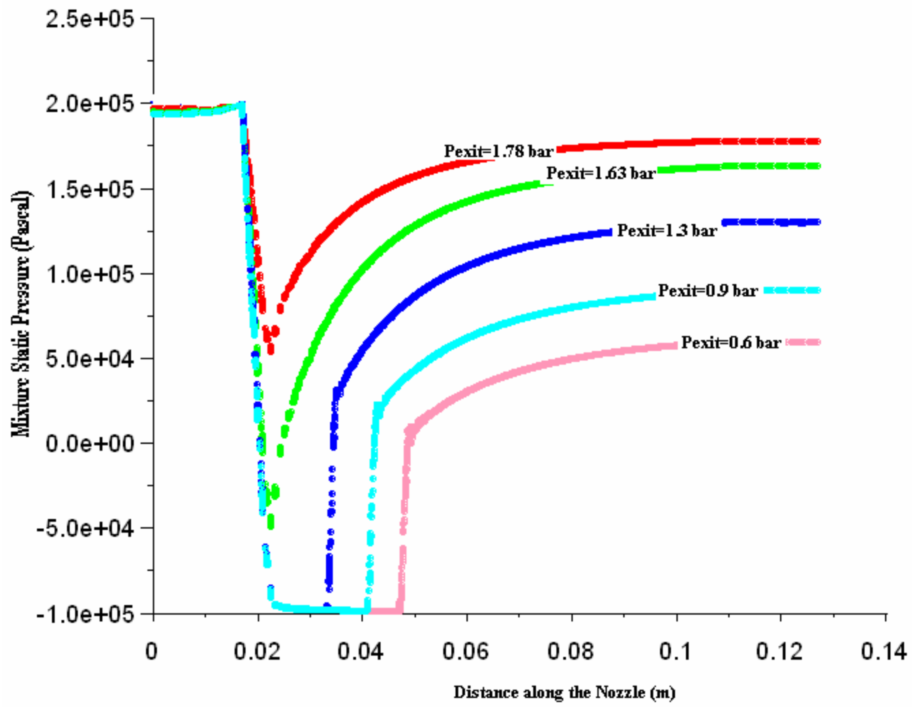

Figure.3a Pressure Distribution Along the Nozzle(table-2a results)

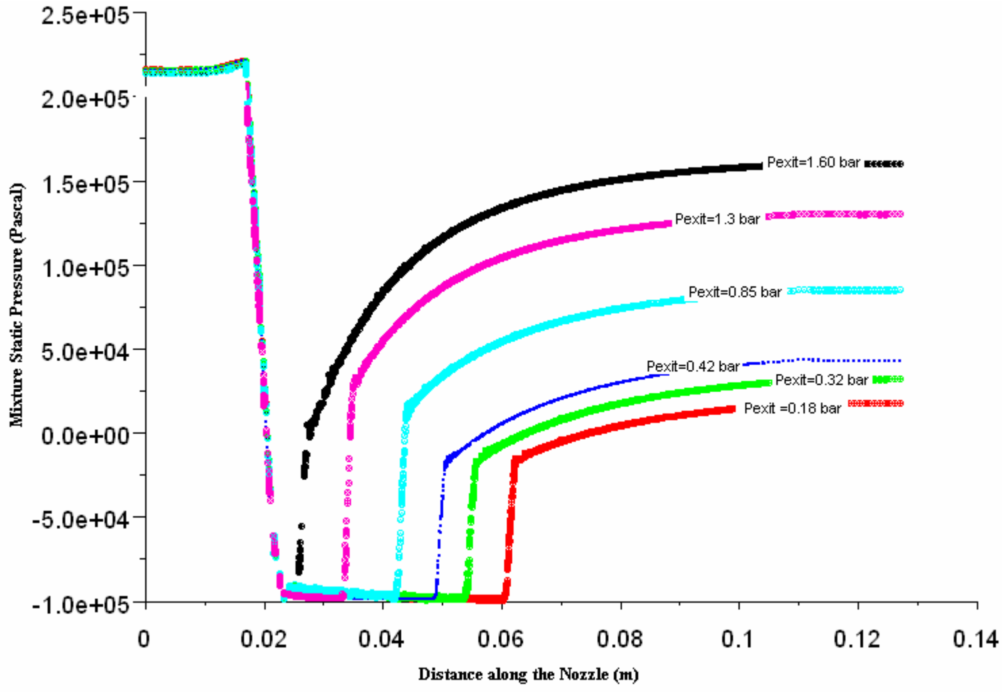

Figure.3b Pressure Distribution Along the Nozzle(table-2b results)

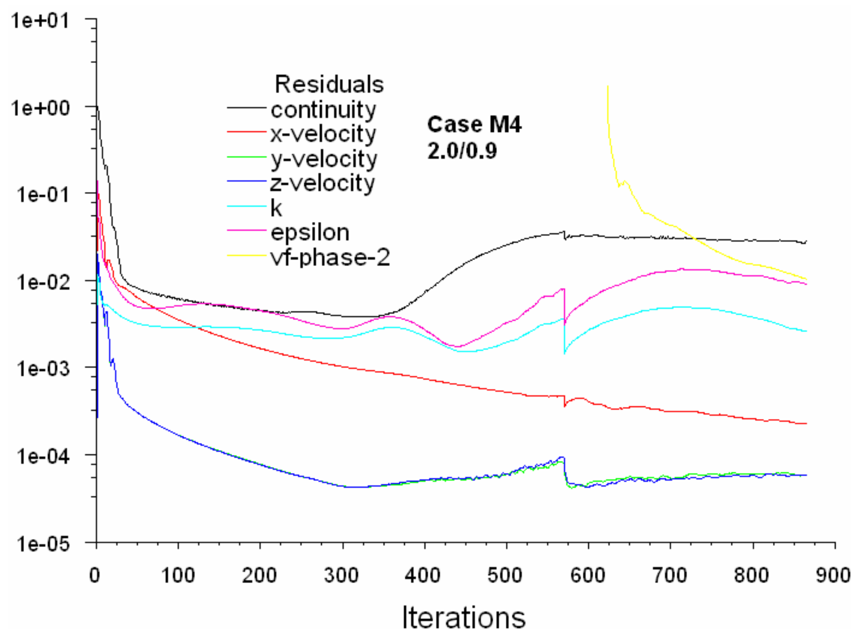

Figure.4 Convergence History(Case M4 Table-2a)

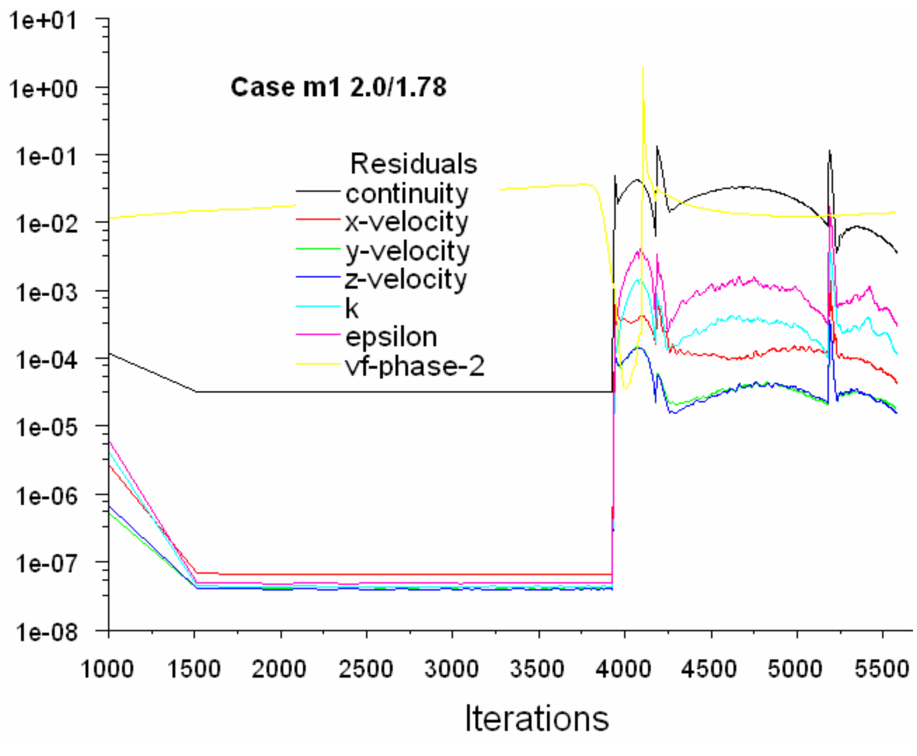

Figure.5 Convergence History (Case M1 table-2a)

دراسة خصائص الجريان ثنائي الطور لنفاث -اقتراب - ابتعاد

د. فياض محمد الدليمي
استاذ مساعد
كليه علوم الحاسبات والرياضيات
قسم الحاسبات

د.غازي يوسف الشاهري
مدرس
قسم الهندسة الميكانيكية
كليه الهندسة

جامعه تكريت

الخلاصة

الدراسة المقدمة تناولت دراسة ظاهرة وجود التكهف من خلال جريان المائع ثنائي الطور الماء والبخار عمليا بأختيار انبوبة فنجوري ثلاثية الابعاد مصنعة من مادة البرسبكس لكونها ملائمة لامكانية رؤية حدوث الظاهرة واجراء قياسات ابعاد منطقة حصول التكهف وتغيير الظروف الحدية (ضغط الدخول والخروج ومعدل التدفق). وباستخدام احد الحقائب الجاهزة (البرامجيات العددية) (FLUENT) لتحديد الظاهرة المَدكورة وهدا يتطلب اختيار الموديل الانسب للجريان الاضطرابي .

التجارب العملية اجريت في مختبرات جامعة كولن الالمانية، طابقت النتائج المستنبطة من الطريقة العددية مع النتائج النظرية التي بينت خصائص الجريان ولاسيما في المناطق التي لايمكن الحصول عليها عمليا من حيث طول منطقة حصول تغير الاطوار اضافة الى الخصائص الاخرى.

الكلمات الدالة

نفاث -أقتراب أبتعاد نموذج أضطرابي تكهف ثنائي الطور