IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 3 Issue 10, October 2016
ISSN (Online) 2348 – 7968 | Impact Factor (2015) - 4.332
www.ijiset.com

Modelling of Shock Waves and Micro Jets Using CFD Analysis

Dr.I.Satyanarayana[1], Shivaji[2],

[1]Principal & Professor of Mechanical Engineering,

Sri Indu institute of Engineering and Technology, sheriguda, Ibrahimpatnam, Ranga Reddy District- 501510

[2]PG student of Mechanical Engineering, Sri Indu institute of Engineering and Technology, sheriguda, Ibrahimpatnam,Ranga Reddy District- 501510

Abstract Main aim of this project is to model shock wave in air and to validate the shock wave properties like Mach number, primary shock strength, reflected shock strength, primary shock wave temperature ratio and reflected shock wave temperature ratios with available theoretical and practical values. Also effect of shock wave properties with variation in parameters like shock tube diameters, shock tube length, viscous and inviscid effects of fluid have been analyzed. Simulated values have well matched with practical data i.e. simulated primary shock strength is matching with theoretical data with an error of less than 10%. In course of simulations Mach number is varied from 2.88 to 7.2, diameters are varied from 10mm to 50mm; lengths are varied from 0.7m to 3.5m for shock tube. From these simulations some of the major conclusions are; maximum Mach number exists till certain length of shock tube and then decreases. There is decrease in shock strength and increase in shock temperature ratios with viscosity effects. In smaller diameter shock tubes, viscous effects are more compared to larger one

. Based on concepts of shock tube, an attempt is made to model micro jets. Micro jets are modelled with fluid as both water and air. Both compressibility and incompressibility effect of fluid has been studied for micro jet model. At last results obtained from micro jet model simulation has been validated with the results available in literature.

***Keywords* – Shock Waves, Mach number, Shock- strength, CFD Analysis, Shock tube, Micro jets, etc.**

1. INTRODUCTION

This project introduces to the exhilarating world of Shock waves and Shock tubes. Shock waves are essentially non-linear waves that propagate at supersonic speeds. Shock wave has unique ability to enhance the pressure and temperature instantaneously in any medium of propagation. Shock waves are being used for many innovative applications in the industry.

Where the shock tube is a device in which a normal shock wave is produced by the sudden bursting of a diaphragm separating a gas at high pressure from one at lower pressure. Shock waves have numerous applications in different fields like aerodynamics, chemical kinetics, medicine, process engineering, drug delivery, agriculture and biology. In aerodynamics shock tubes are used to generate shock waves of required strength in a constant area tube. Shock tube is a very valuable tool for the study of physical and chemical processes at high temperature.

This project also introduces to high-speed Micro jets. Micro jets are fluids of diameter 10 micro meter to 300 micro meter with velocity of jet varying from 10 m/s to 2000 m/s. Micro jets have many applications in fluid dynamics such as micro propulsion, biomedical application like needle less drug delivery, engineering applications like to introduce Shocks in the combustion ducts of aircraft engines etc.

2. LITERTURE REVIEW

By. O.E. Kosing and B.W. Skews:[1] At the end of last century the high speed metal forming technique came in picture. Generation of liquid shock waves and the behavior of circular metal plates have experimentally done by O.E. Kosing and B.W. Skews at The University of the Witwatersrand, School of Mechanical Engineering, Flow Research Group, Johannesburg, South Africa. By. Broullette:[4] The flow field in conventional shock tube was extensively studied and documented in many of the past work. But as flow dimension reduces the Reynolds number decreases and leads to more prominent viscous effects. This leads to attenuation of shock strength and eventually leads to less propagation distance .Shock wave also suffers attenuation as the initial operating pressure reduces. This has been pointed out by the experimental studies conducted on a shock tube of 5.3mm diameter

3. PROCEDURE FOR CFD SOLUTION

CFD codes are structured around the numerical algorithms that can be tackle fluid problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces input problem parameters and to examine the results. It includes mainly three steps which has been shown in fig1: below

Fig2:Laboratory Model of Shock Tube

- Pre-Processing (Gambit)
- Solver (Fluent)
- Post-Processing (Fluent)

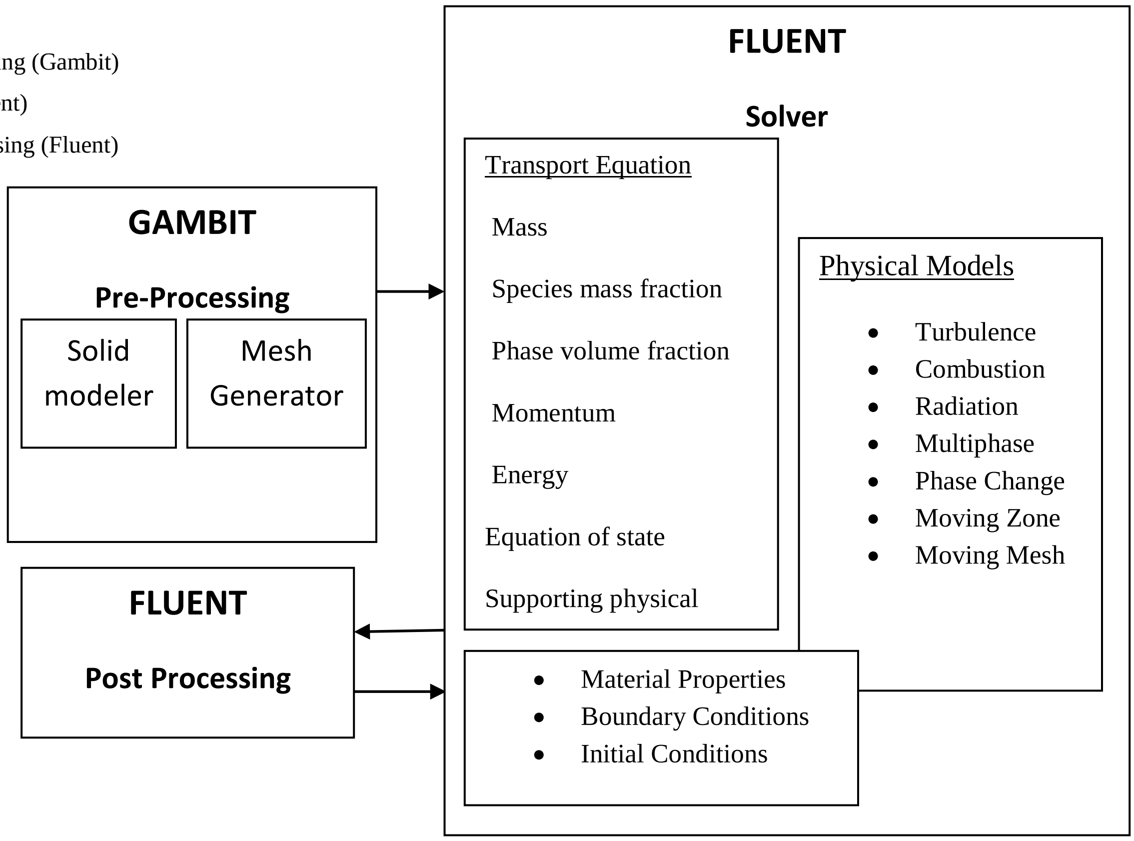

Fig1 :CFD frame work

4. DESIGN OF SHOCK TUBE

One of the design models of this project is shock

Mesh is applied to the created geometry. After creation of mesh, the model will look like as in (fig3) below

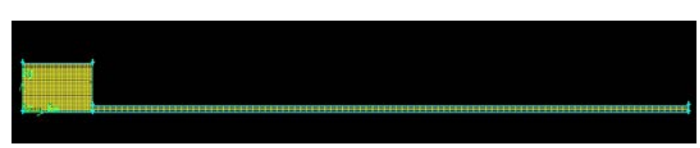

Fig 3: Basic Geometry of Axis Symmetric Model of Shock Tube Meshed in GAMBIT

After applying boundary conditions, save the meshed geometry file and export it as a 2d mesh file (as model is two dimensional).

IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 3 Issue 10, October 2016
ISSN (Online) 2348 – 7968 | Impact Factor (2015) - 4.332
www.ijiset.com

After this step importing the above meshed file into 2ddp version of Fluent, setting dynamic mesh parameters, setting fluid properties, setting solution control panel, initializing and iterating to solve the generated model.

At last in post-processing step, displaying the contours of pressure and Mach number (as shown below in fig 4,5,&6) to analyze the results obtained by simulation.

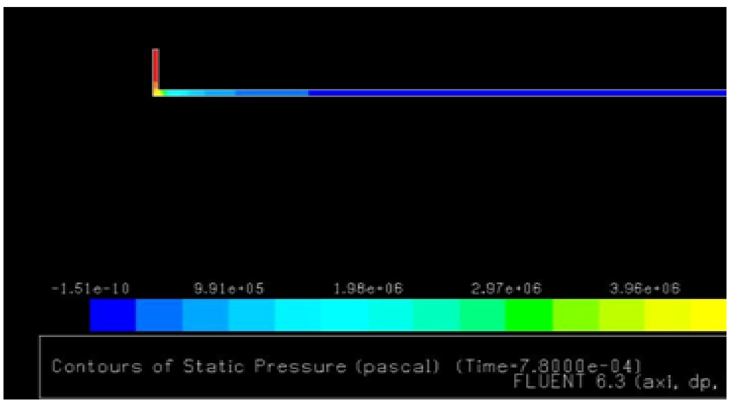

Fig 4: Contours of Static Pressure in Driver Section

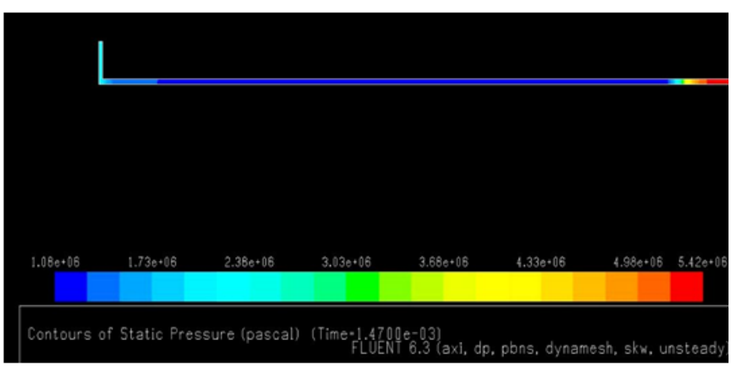

Fig 5: Contours of Static Pressure in Driven Section

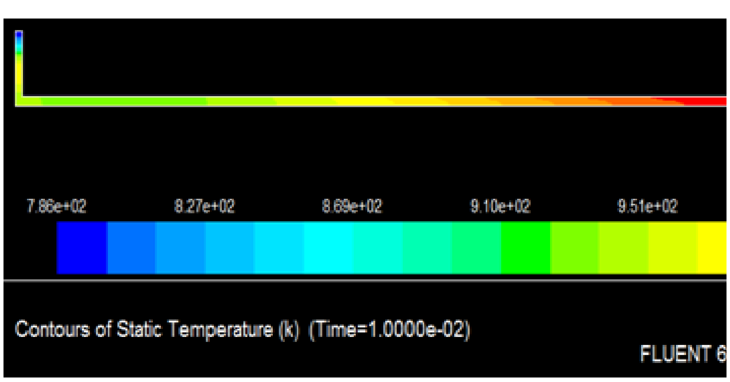

Fig 6: Contours of Static Temperature

5. DESIGN OF MICRO JETS

Laboratory model of micro jet is shown in fig6.

A CFD model of micro jet is created with the following dimensions (refer fig12).D=3mm, d=0.3mm, L=3.5mm, l=0.5mm. for a diaphragm velocity of 1m/s.

Fig7: Laboratory Model of Micro Jet

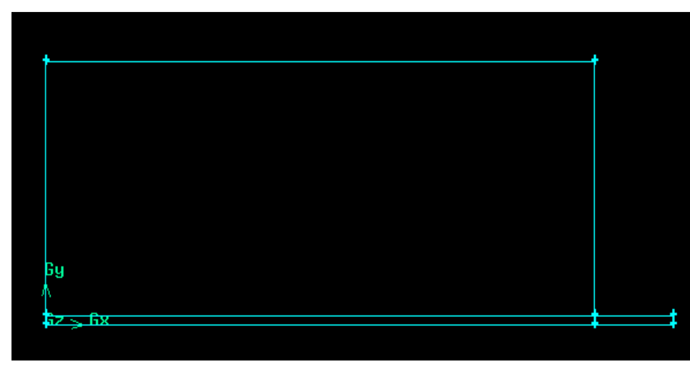

Fig8: Basic Geometry Created for Axis Symmetric Model of Micro Jet in GAMBIT

After this step, mesh is applied to the above created geometry. After creation of mesh, the model will look like as below.

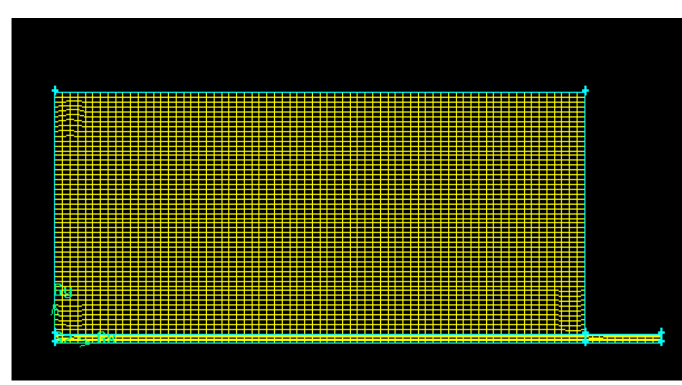

Fig9: Basic Geometry Meshed in GAMBIT

After applying boundary conditions, save the meshed geometry file and export it as a 2d mesh file (as model is two dimensional).

After this step importing the above meshed file into 2ddp version of Fluent, setting dynamic mesh parameters, setting fluid properties, setting solution control panel, initializing and iterating to solve the generated model.

At last in post-processing step, displaying the contours of static pressure, velocity and temperature etc can be displayed and analysed like as shown below

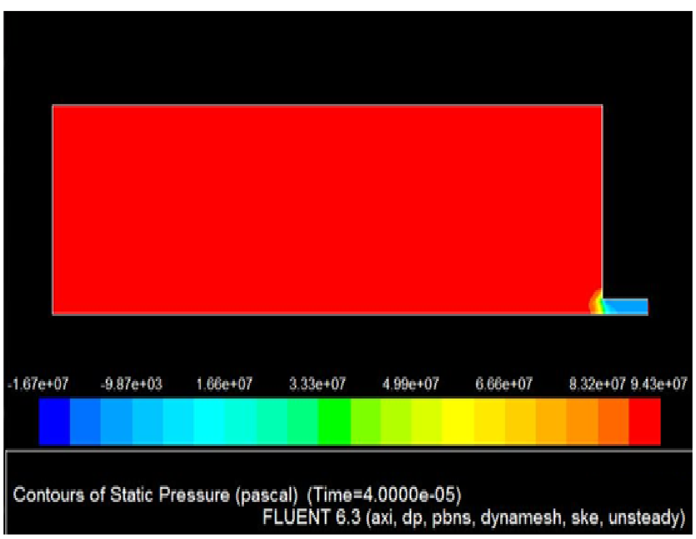

Fig 10: Contours of Static Pressure in Micro Jet Mode

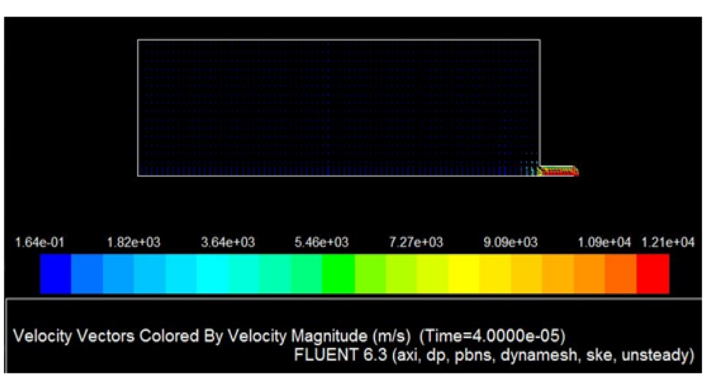

Fig 11: Vector Plot of Velocity in Micro Jet Model

6.1. RESULTS AND DISCUSSIONS OF SHOCK TUBE

Effect on Shock Wave Properties with Changing $\frac{P4}{P1}$ (Inviscid Model)

The Mach number is a direct function of driver pressure ratio $(\frac{P4}{P1})$ which is as shown in fig 21, studied the all variations with respect to the Mach number than driver pressure ratio. The following results are shown in the figures (i.e. fig 21 to fig 25) below.

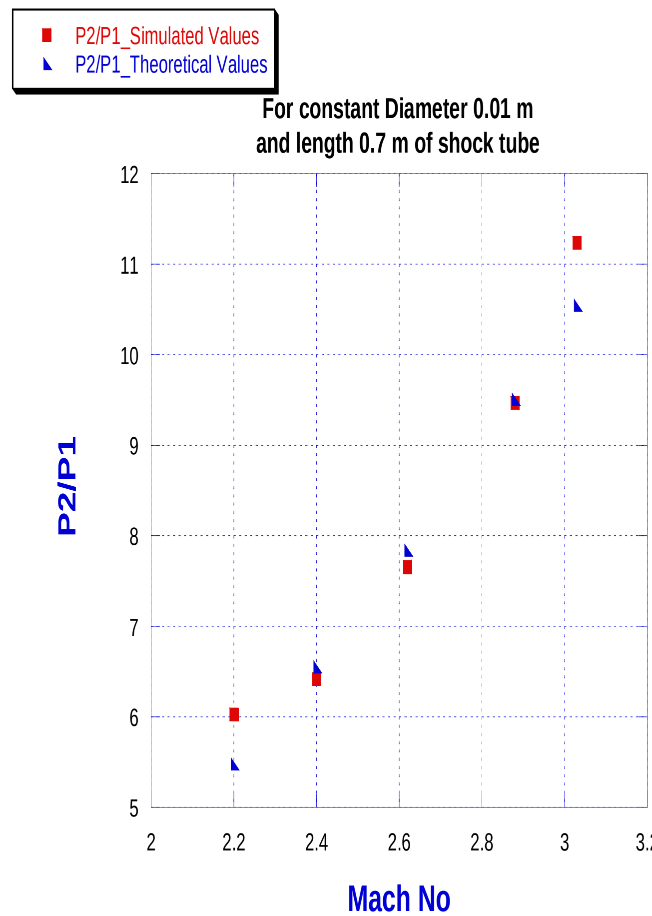

Fig 12: Mach No. Vs Primary Shock Strength

Table 5.2: Effect on Shock wave Properties with Changing P4/P1

% Compression	Mach No	Simulated Values					Theoretical Values			
		P2/P1	P5/P1	P4/P1	T2/T1	T5/T1	T2/T1	T5/T1	P2/P1	P5/P1
88	2.2	6.03	23.2	14.42	1.86	2.86	1.86	2.88	5.48	20.45
90	2.41	5.40	25.34	16.8	1.93	2.99	2.04	3.32	5.61	26.85
92	2.62	7.66	35.05	23.42	2.13	3.36	2.54	3.80	7.84	34.97
94	2.88	9.47	48.06	37.86	2.38	3.90	2.92	4.44	9.51	46.04
96	3.03	11.24	61.79	49.05	2.63	4.36	2.71	4.83	10.54	49.00

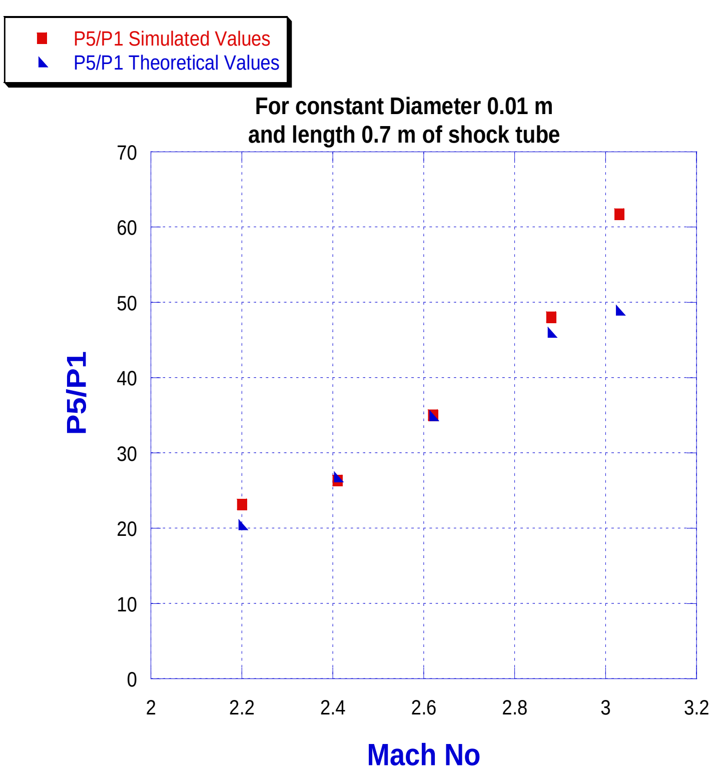

Fig 13: Mach No. Vs Reflected Shock Strength

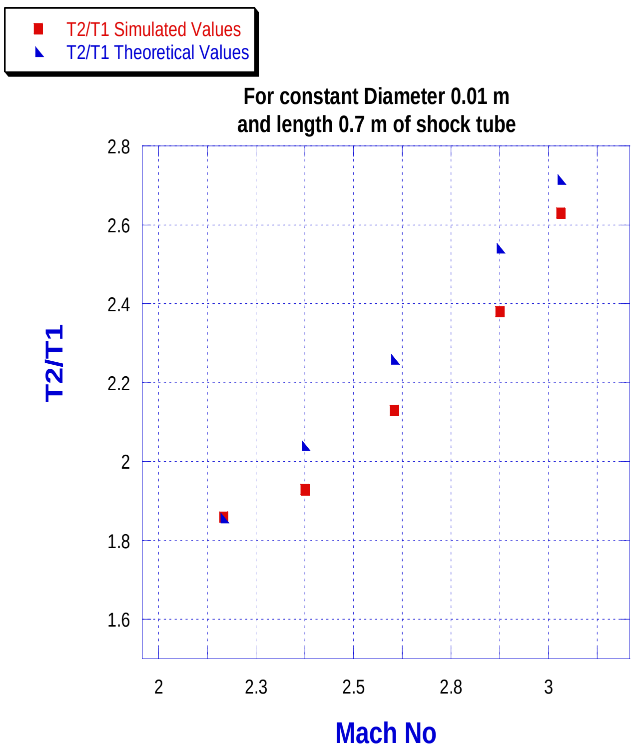

Fig 14: Mach No. Vs Primary Shock Temperature

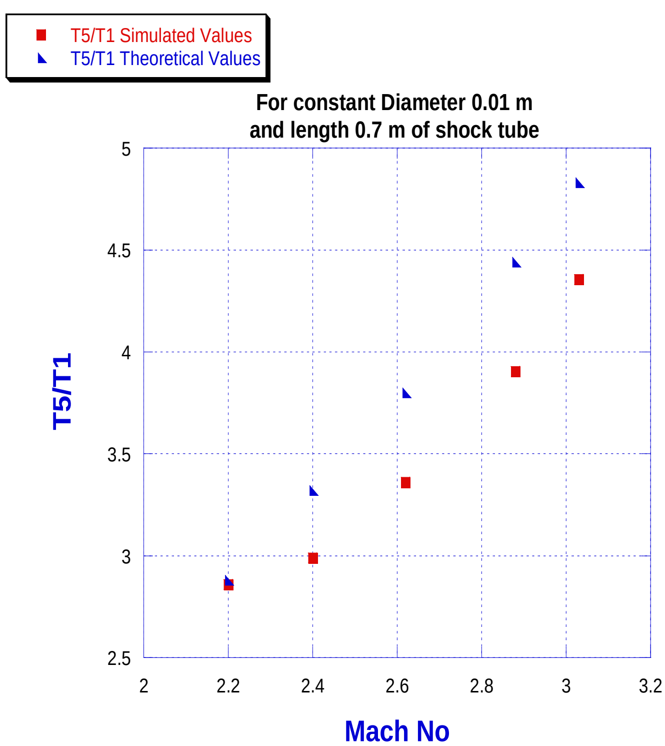

Fig 15: Mach No. Vs Reflected Shock Temperature

From the above graphs (fig21 to fig25), it can be conclude that, the primary shock strength $(\frac{P2}{P1})$, reflected shock strength $(\frac{P5}{P1})$, primary shock temperature ratio $(\frac{T2}{T1})$ and reflected shock temperature ratio $(\frac{T5}{T1})$ are direct function of Mach number and there are some variations in simulation results with theoretical results because the mass flow not instantaneous.

The variation occurred in simulated values of primary shock strength $(\frac{P2}{P1})$, reflected shock strength $(\frac{P5}{P1})$, primary shock temperature ratio $(\frac{T2}{T1})$ and reflected shock temperature ratio $(\frac{T5}{T1})$ on comparison with the theoretical values are less than are equal to 10%, 26%, 18% and 10% respectively.

6.2. RESULTS AND DISCUSSIONS OF MICRO JETS

For a diaphragm velocity of 1 m/sec the fluid velocity at outlet is 120m/sec (refer fig 26) which is matching with the practical data with an error of 16.67%.

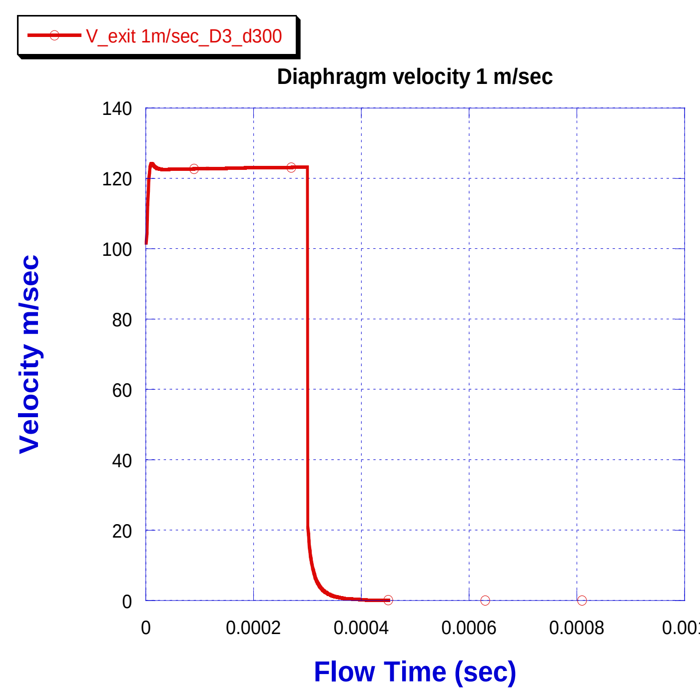

Fig 16: Velocity of Fluid at the Exit for a Diaphragm Velocity of 1m/sec

A CFD model is created with the following dimensions

D = 3mm, d = 0.1mm, L = 3.5mm and l = 0.5mm (refer fig27) by considering air as fluid medium and simulations are done. Air is considered as compressible fluid and a velocity of 2m/s is given to the diaphragm which shows velocity of fluid 160m/s at exit,

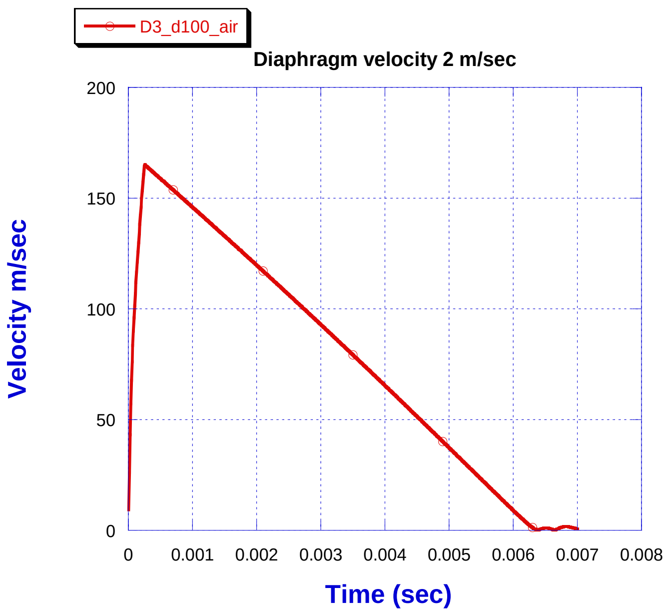

Fig 17: Velocity of Fluid for Compressible Air at Exit

Velocity of Fluid at Exit for Variable Inlet Diameter

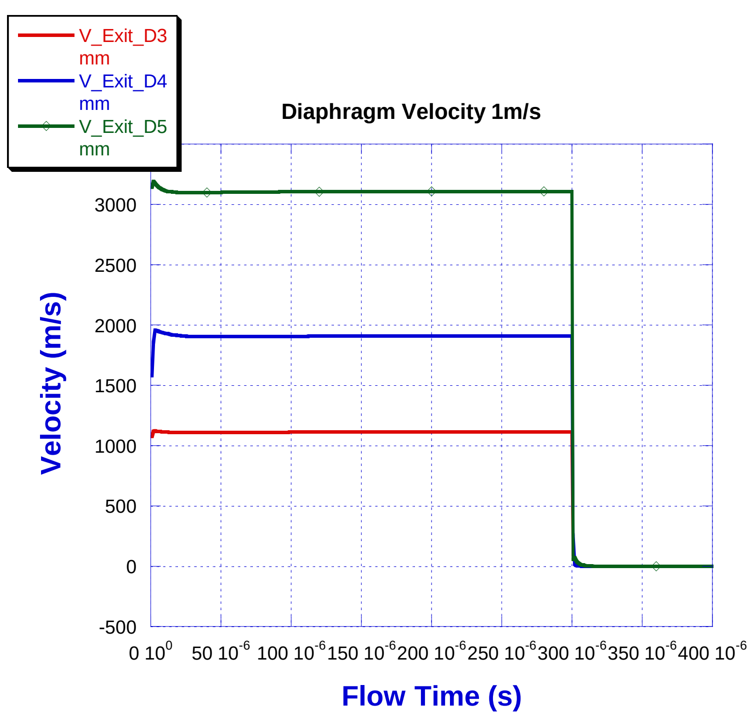

Fig 18: Velocity of Fluid at Exit for Variable Inlet Diameter

From the (fig 28) it can be observed that the velocity of fluid is proportional (not linear proportional) to initial diameter i.e. increase in initial diameter will lead to increase in fluid at exit because of increased mass flow

7. CONCLUSIONS

7.1 Conclusions on Shock Tube Model

1. Mach number (M) is a function of Initial reservoir pressure ratio (P_4/P_1), as reservoir pressure increases, Mach number also increases.
2. Primary shock strength (P_2/P_1) is a function of Mach number(M), as Mach number increases the shock strength also increases.
3. Reflected shock strength (P_5/P_1) is a function of Mach number(M), as mach number increases the reflected shock strength also increases.
4. Primary shock temperature ratio (T_2/T_1) is function of Mach number(M), as Mach number increases the primary shock temperature ratio also increases.
5. Reflected Shock temperature ratio (T_5/T_1) is function of Mach number (M), as Mach number increases the reflected shock temperature ratio also increases.
6. Mach number for inviscid model is more than viscous fluid model. It is obvious because velocity of shock is less in viscous medium.
7. As Mach number is less in viscous fluid, hence shock strength (P_2/P_1) and reflected shock strength (P_5/P_1) are also less in viscous fluid than in inviscid fluid.
8. Primary shock temperature ratio (T_2/T_1) and reflected Shock temperature ratio (T_5/T_1) is more in viscous medium than inviscid medium. It is obvious because of viscosity, viscous fluid gets heated more than inviscid fluid.
9. For constant reservoir pressure and diameter of shock tube, as length of shock tube increases to its certain value, the shock strength also increases and hence Mach number, but again the shock strength decreases as length of shock tube exceeds the certain length. But, primary and reflected shock temperature ratio goes on increasing; it is because of continuous viscous heating of fluid, as shock propagates forward.
10. For constant reservoir pressure and length of shock tube, as diameter of shock tube increases, because of less viscous effect, the shock strength increases. Viscous effect is more for smaller diameter of shock tube, so shock speed is less and hence Mach no is less.

7.2 Conclusions on Micro Jet Model

1. Velocity of jet depends upon outlet diameter of cylinder, as outlet diameter decreases, velocity of jet increases.

2. Velocity of jet also depends upon inlet diameter, as inlet diameter of cylinder increases, hence mass inlet increases, velocity of jet also increases.
3. Velocity of jet does not depend much upon the flow length of cylinder, as length of cylinder increases.

8. REFERENCES

[1] O.E. Kosing and B.W. Skews, July 20-25, 1997, "The use of liquid shock waves for metal forming," *21st International Symposium on Shock Waves (ISSW 21), Great Keppel Island, Australia*, Paper 5289.

[2] Clemdson CJ, Hultmon HI, Air embolism and the cause of death in blast

Injury,mil surg 1954; 114: 424-437

[3] Morrissey,M.M. and B. Choute (1997), Burst conditions of Explosive

Volcanic eruptions recorded on microbarographs, science, 275(5304),

1290-1293

[4] M. Brouillette, "Shock Waves at microscales" shock waves, Vol, 13,

Vol,13,No.1, 2003, pp. 3-12

[5] D.E. Zeitoun.Y. Burstshell et al . Numerical simulation of shock wave

Propagation in microchannels using continuum and kinetic approaches

Shock Waves, Vol. 19, No. 4, 2009, pp. 307-316

[6] Yusoff. M. Z. A two dimensional Time Accurate Euler solver for Turbo

Machinery Application Journal-Institution of engineers Malaysia.Vol.5

No. 3, 1998.

[7] M.A.F. Kendall, N.J. Quinlan,R.W. Thorpe, S. J. Anisworth, and B.J.-

Bellhouse. Measurements of the gas and particle flow within a

Convergent-Divergent nozzle for high speed powdered vaccine and drug

delivery, Experiments in fluids Vol. 37(2004) 128-136

[8] Y. Liu and M.A.F. Kendall, Numerical analysis of gas and micro particle

Interactions in a hand held shock- tube device, Biomed devices, Vol.8

(2008) 341-351

[9] Y.S. Weber. The numerical simulation of the reflected-shock/boundary-

layer interaction in shock tubes. Ph. D. thesis, university of the

Maryland college park, 1994

[10] B. Sjogreen. H.C. Yee, Greed convergence of high order methods for

multiscale complex unsteady viscous compressible flows, J. Comput,

phys185(2003) 1-23

[11] Gas Dynamics' 3rd Edition by Ethirajan Rathakrishnan.

[12] Date A. W., "Introduction to Computational Fluid Dynamics", 1st Edition

Cambridge University Press, 2005

[13] Gaydon A.G. and Harle I.R., "Shock Tube in High-Temperature

Chemical Physics", Reinhold Publishing, NewYork, 1963.

[14] Miiller, M., StoBwellenfokussierung in Wasser, Dissertation, RWTH

Aachen, Germany, pp 14-15, 1987

[15] Wilson, F.W., High-velocity forming of metals, National Technical Publications Committee, American Society of Tool and Manufacturing Engineers, Prentice-Hall INC., pp. 162- 181, 1964