

# Designing and Simulating Compressible Flow in a Nozzle

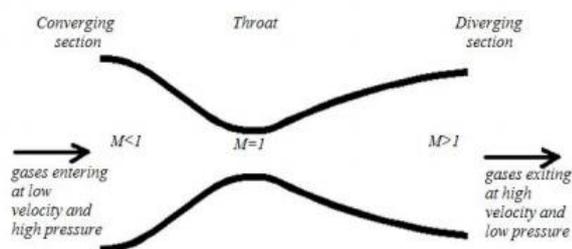
V. Tapasvi, M. Satyanarayana Gupta, T. Kumaraswamy

**Abstract**— Compressible flow is the branch of fluid mechanics that deals with flows having significant changes in fluid density. Gases, but not liquids, display such behavior. To distinguish between compressible and incompressible flow in gases, the Mach number must be greater than about 0.3 before significant compressibility occurs. A nozzle is a device designed to control the direction or characteristics of a fluid flow (especially to increase velocity) as it exits (or enters) an enclosed chamber or pipe. Now we are designing a nozzle by using a gambit design software and then converting that into a ansys software for analysis .In that analysis we are giving boundary conditions and inlet and outlet. This analysis totally on c-d nozzle. By using supersonic steam

**Keywords**:- fluid mechanics, density, incompressible flow, Mach number, compressibility, fluid flow.

## I. INTRODUCTION

A nozzle is capable of transforming pressure energy into kinetic energy and vice versa, Depending on shape of the nozzle above transformations occur. Nozzle which is a C-D nozzle is used to attain supersonic flow speeds. An ideal nozzle is depicted in the figure. The inlet mach number is less than one, Convergent section accelerates it to sonic velocity at the throat and further accelerated to supersonic velocities by the diverging section.



A **nozzle** is a device designed to control the direction or characteristics of a fluid flow (especially to increase velocity) as it exits (or enters) an enclosed chamber or pipe. A nozzle is often a pipe or tube of varying cross sectional area, and it can be used to direct or modify the flow of a fluid (liquid or gas). Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them. In nozzle velocity of fluid increases on the expense of its pressure energy.

**Manuscript published on 30 August 2015.**

\* Correspondence Author (s)

**V. Tapasvi**, Department of Aeronautical, Jawaharlal Nehru Technological University, Hyderabad, India.

**Dr. M. Satyanarayana Gupta**, Department of Aeronautical, Jawaharlal Nehru Technological University, Hyderabad, India.

**T. Kumaraswamy**, Department of Aeronautical, Jawaharlal Nehru Technological University, Hyderabad, India.

© The Authors. Published by Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP). This is an [open access](http://creativecommons.org/licenses/by-nc-nd/4.0/) article under the CC-BY-NC-ND license <http://creativecommons.org/licenses/by-nc-nd/4.0/>

## II. INTRODUCTION TO COMPUTATIONAL FLUID DYNAMICS (CFD)

Advances in computing technology, software and hardware have revolutionized the design process of engineering vehicles such as aircrafts, automobiles and ships. Many commercial software packages are being used in the design as well as analysis processes which not only save the lead time and costs of new designs, but also are used to study systems where controlled experiments are difficult or impossible to perform. In the area of fluid dynamics, there are many commercial Computational Fluid Dynamics (CFD) packages available for modeling flow in or around objects. Computational Fluid Dynamics (CFD) has been constantly developed over the past few decades and now both commercial and research codes can provide more and more robust and accurate results. Combined with the use of wind tunnel test data, CFD can be used in the design process to drive geometry changed instead of being used mainly as a design validation tool. Computational Fluid Dynamics (CFD) has become an integral part of the engineering design and analysis environment of many companies that require the availability to predict performance of new designs or processes before they are ever manufactured or implemented. One of the most critical requirements for any CFD tool used for thermal applications is the ability to simulate flows along nozzles, turbines. Such challenging features has pressure gradients, shocks, velocity distribution, eddy location, stream line curvature, and stream wise vortices pose a challenge for computing. The small margins of improvement that are usually targeted in nozzle and turbines design today require precise tools capable of discerning small differences between alternate designs. Custom modeling tools that are based as simplified numerical methods and assumptions cannot provide accuracy that can be obtained with CFD, which offers mainly inherent advantages for ex: it offers quick and cheap solution and comparison to experimental solution and more accurate in comparison to empirical methods used in design. Accurate simulation of flows through the nozzle is important for prediction of velocity pattern and pressure pattern. Fluid dynamics deals with the dynamic behavior of fluids and its mathematical interpretation is called as Computational Fluid Dynamics. Fluid dynamics is governed by sets of partial differential equations, which in most cases are difficult or rather impossible to obtain analytical solution. CFD is a computational technology that enables the study of dynamics of things that flow. The Physical aspects of any fluid flow are governed by three fundamental principles: Mass is conserved; Newton's second law and Energy is conserved.

These fundamental principles can be expressed in terms of mathematical equations, which in their most general form are usually partial differential equations. Computational Fluid Dynamics is the science of determining a numerical solution to the governing equations of fluid flow whilst advancing the solution through space or time to obtain a numerical description of the complete flow field of interest. Computational Fluid Dynamics thus provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of Mathematical modeling (partial differential equations) Numerical methods (discretisation and solution techniques). Software tools (solvers, pre- and post processing utilities) CFD enables scientists and engineers to perform 'numerical experiments' (i.e. Computer simulations) in a 'virtual flow laboratory' real experiment CFD simulation. The procedure for the CFD analysis in FLUENT follows the simple steps below:

- i. The model used for the analysis is drawn in, Ansys design modeller and meshed in the pre-processor Ansys meshing, which is the compatible modelling software for FLUENT. All the files for the geometry and meshing of the model are saved as mesh or grid file.
- ii. Next, in FLUENT, the saved mesh or grid file of the model is read, checked and scaled for the required working unit.
- iii. The model is defined for the type of solver and boundary conditions to be used. The model is defined according to the type of analysis required in the research project.
- iv. The model is solved by setting the required parameters in the solution panel and then iterated for convergence.
- v. Results can be obtained from the graphic display and report in FLUENT. Results can be displayed in terms of contour, velocity vector, and particle track and path line. Any calculation required can be performed in FLUENT also.
- vi. Finally, the results and all the data can be saved for future references by writing the files.

Next, the following section will discuss on the governing equations used in FLUENT when computing and analyzing the fluid flow behaviour. All the equations will give the details on how the CFD works in order to simulate the result for certain problems.

### III. GOVERNING EQUATIONS IN CFD

There are mainly three equations we solve in computational fluid dynamics problem. They are Continuity equation, Momentum equation (Navier Stokes equation) and Energy equation. The flow of most fluids may be analyzed mathematically by the use of two equations. The first, often referred to as the *Continuity Equation*, requires that the mass of fluid entering a fixed control volume either leaves that volume or accumulates within it. It is thus a "mass balance" requirement posed in mathematical form, and is a scalar equation. The other governing equation is the *Momentum Equation*, or *Navier-Stokes Equation*, and may be thought of as a "momentum balance". The Navier-Stokes equations are vector equations, meaning that there is a separate equation for each of the coordinate directions (usually three).

### IV. CONTINUITY EQUATION

The equation of continuity expresses the conservation of matter--if matter flows away from a point, there must be a decrease in the quantity remaining. By definition, the continuity equation should be recognized as a statement of mass conservation. The Continuity Equation relates the speed of a fluid moving through a pipe to the cross sectional area of the pipe. It defines that as a radius of the pipe decreases the speed of fluid flow must increase and vice-versa. A continuity equation expresses a conservation law by "Equating a net flux over a surface with a loss or gain of material within the surface". Continuity equations often can be expressed in either integral or differential form as shown below [4.1-4.5].

$$\int_{cs} \rho V dA + \frac{\partial}{\partial t} \int_{cv} \rho dA = 0 \dots\dots\dots (4.1)$$

This is a statement of the principle of mass conservation for a steady, one-dimensional flow, with one inlet and one outlet.

$$\nabla(\rho V) + \frac{\partial \rho}{\partial t} = 0 \dots\dots\dots (4.2)$$

Where,

$$\nabla = \frac{\partial}{\partial x} \hat{i} + \frac{\partial}{\partial y} \hat{j} + \frac{\partial}{\partial z} \hat{k}$$

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho \cdot u)}{\partial x} + \frac{\partial(\rho \cdot v)}{\partial y} + \frac{\partial(\rho \cdot w)}{\partial z} = 0 \dots\dots\dots (4.3)$$

#### *i. Momentum (Navier Stokes) Equations*

The momentum equation is a statement of Newton's Second Law and relates the sum of the forces acting on an element of fluid to its acceleration or rate of change of momentum. The Newton's second law of motion  $F = ma$  forms the basis of the momentum equation. In fluid mechanics it is not clear what mass of moving fluid we should use so we use a different form of the equation. The Navier-Stokes equations are the fundamental partial differentials equations that describe the flow of incompressible fluids. In fluid dynamics, the **Navier-Stokes equations** are a set of nonlinear partial differential equations that describe the flow of fluids such as liquids and gases. The equations are a set of coupled differential equations and could, in theory, be solved for a given flow problem by using methods from calculus. The Navier-Stokes equations consist of a time-dependent continuity equation for conservation of mass, three time-dependent conservation of momentum equations and a time-dependent conservation of energy equation. The NSE is regarded as the ultimate answer to fluid dynamic problems. These equations may as well be the most widely studied equations in applied physics. The range of validity of the Navier-Stokes is only limited by the model used for the viscous stresses. There are thus three different momentum equations that together comprise the Navier-Stokes Equations (4.4)



$$\begin{aligned} \rho \frac{\partial u}{\partial t} + \rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} + \rho w \frac{\partial u}{\partial z} &= \rho g_x - \frac{\partial p}{\partial x} + \mu \frac{\partial^2 u}{\partial x^2} + \mu \frac{\partial^2 u}{\partial y^2} + \mu \frac{\partial^2 u}{\partial z^2} \\ \rho \frac{\partial v}{\partial t} + \rho u \frac{\partial v}{\partial x} + \rho v \frac{\partial v}{\partial y} + \rho w \frac{\partial v}{\partial z} &= \rho g_y - \frac{\partial p}{\partial y} + \mu \frac{\partial^2 v}{\partial x^2} + \mu \frac{\partial^2 v}{\partial y^2} + \mu \frac{\partial^2 v}{\partial z^2} \\ \rho \frac{\partial w}{\partial t} + \rho u \frac{\partial w}{\partial x} + \rho v \frac{\partial w}{\partial y} + \rho w \frac{\partial w}{\partial z} &= \rho g_z - \frac{\partial p}{\partial z} + \mu \frac{\partial^2 w}{\partial x^2} + \mu \frac{\partial^2 w}{\partial y^2} + \mu \frac{\partial^2 w}{\partial z^2} \end{aligned}$$

(4.4)

ii. Energy Equation

$$\begin{aligned} \frac{\partial}{\partial t} \left( \rho e + \frac{1}{2} \rho v^2 \right) + \frac{\partial}{\partial x} \left( \rho u e + \frac{1}{2} \rho u v^2 \right) + \frac{\partial}{\partial y} \left( \rho v e + \frac{1}{2} \rho v v^2 \right) + \frac{\partial}{\partial z} \left( \rho w e + \frac{1}{2} \rho w v^2 \right) &= \\ k \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) - \left( u \frac{\partial p}{\partial x} + v \frac{\partial p}{\partial y} + w \frac{\partial p}{\partial z} \right) + \\ \mu \left[ u \frac{\partial^2 u}{\partial x^2} + \frac{\partial}{\partial x} \left( v \frac{\partial v}{\partial x} + w \frac{\partial w}{\partial x} \right) + v \frac{\partial^2 u}{\partial y^2} + \frac{\partial}{\partial y} \left( u \frac{\partial u}{\partial y} + w \frac{\partial w}{\partial y} \right) + w \frac{\partial^2 u}{\partial z^2} + \frac{\partial}{\partial z} \left( u \frac{\partial u}{\partial z} + v \frac{\partial v}{\partial z} \right) \right] \\ + 2\mu \left[ \frac{\partial^2 u}{\partial x^2} + \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial v}{\partial z} \frac{\partial w}{\partial y} + \frac{\partial^2 w}{\partial z^2} + \frac{\partial w}{\partial x} \frac{\partial u}{\partial z} \right] + \rho u g_x + \rho v g_y + \rho w g_z \end{aligned}$$

(4.5)

The energy equation is a scalar equation, meaning that it has no particular direction associated with it. This equation demonstrates that, per unit volume, the change in energy of the fluid moving through a control volume is equal to the rate of heat transferred into the control volume plus the rate of work done by surface forces plus the rate of work done by gravity. This expression of the energy equation is valid for most applications. Mathematically, the energy equation is given in eq(4.5).

A. Introduction to Fluent

ANSYS Fluent software contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants. Special models that give the software the ability to model in-cylinder combustion, aero acoustics, turbo machinery, and multiphase systems have served to broaden its reach. Fluent uses finite volume numerical procedures to solve the governing equations for fluid velocities, mass flow, pressure, temperature, species concentration and turbulence parameters and fluid properties. Numerical techniques involve the sub-division of the domain into a finite set of neighboring cells known as "control volumes" and applying the discretized governing partial differential equations over each cell. This yields a large set of simultaneous algebraic equations, which are highly non-linear. These equations are in turn solved by iterative means until a converged solution is achieved. The criteria of convergence can be changed by the user, and is generally applied to the changes in the values of all the field variables from one iteration to the next. When all the equations are satisfied on all the discretization points there will be no change from one iteration to the next. This theoretical convergence is not normally achievable in a finite number of

steps. Hence the selection of suitable criteria to detect near convergence becomes important.

B. Solver Settings

The solver chosen for this analysis is pressure-based unsteady solver. The pressure-based approach was developed for low-speed incompressible flows also compressible flows with lower Mach number, while the density-based approach was mainly used for high-speed compressible flows. However, recently both methods have been extended and reformulated to solve and operate for a wide range of flow conditions beyond their traditional or original intent. In the pressure-based approach, the pressure field is extracted by solving a pressure or pressure correction equation which is obtained by manipulating continuity and momentum equations. The pressure-based solver uses a solution algorithm where the governing equations are solved sequentially (i.e., segregated from one another). Because the governing equations are non-linear and coupled, the solution loop must be carried out iteratively in order to obtain a converged numerical solution. In the segregated algorithm, the individual governing equations for the solution variables are solved one after another. Each governing equation, while being solved, is "decoupled" or "segregated" from other equations, hence its name. The segregated algorithm is memory-efficient, since the discretized equations need only be stored in the memory one at a time. However, the solution convergence is relatively slow, inasmuch as the equations are solved in a decoupled manner. With the segregated algorithm, each iteration consists of the steps illustrated in below Fig 4.1.

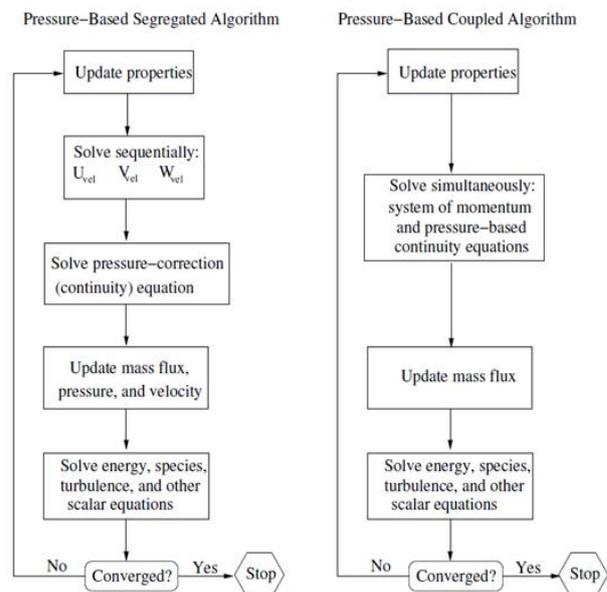


Fig. 4.1: Overview of the Pressure-Based Solution Methods

- Update fluid properties (e.g., density, viscosity, specific heat) including turbulent viscosity (diffusivity) based on the current solution.
- Solve the momentum equations, one after another, using the recently updated values of pressure and face mass fluxes.



- Solve the pressure correction equation using the recently obtained velocity field and the mass-flux.
- Correct face mass fluxes, pressure, and the velocity field using the pressure correction obtained from Step 3.
- Solve the equations for additional scalars, if any, such as turbulent quantities, energy, species, and radiation intensity using the current values of the solution variables.
- Update the source terms arising from the interactions among different phases (e.g., source term for the carrier phase due to discrete particles).
- Check for the convergence of the equations.

### C. Viscous Flows

When talking about fluids, two different states of flows exist and they are easily identified: laminar flow and turbulent flow. Laminar flows are the ones that smoothly vary their velocity fields in space and time in which individual “sheets” of fluid move past each other without generating cross currents. This type of flows appear when the fluid viscosity force is big, in comparison with the inertial forces, and they damp out perturbations that may happen because of imperfections and irregularities. These flows occur at low values of the Reynolds number:

$$Re = \frac{\rho V L}{\mu} \dots\dots 4.6$$

where L is a characteristic length

The other hand, turbulent flows are characterized by big fluctuations in velocity and pressure in space and time, sometimes nearly at random. These flows have fluctuating velocity fields. The fluctuations turn up from instabilities that grow until some interactions make the fluctuations split into smaller and smaller whirlwinds that dissipate in the end (generally by heat formation) due to the action of viscosity. These flows take place at high Reynolds numbers. The fluctuations can be of small size and high frequency, so it is very cost effective to make simulations of them directly in practical engineering cases. What it is done is that the equations of behaving can be ensemble-averaged, time averaged, or small scales can be removed. The equations after these modifications are easier to solve; nevertheless, the modifications add other incognita and different turbulence models appear to determine these unknown variables. FLUENT have the following turbulence models:

- Spalart- Allmaras model.
- k-epsilon models (k-ε).
- Standard k- ε model.
- Renormalization-group (RNG) k- ε model.
- Realizable k- ε model.

k-omega models (k-ω)

- Standard k-ω model.
- Shear-stress transport (SST) k-ω model.
- v2 - f model.
- Reynolds stress model (RSM).

### D. Turbulence Models In FLUENT

‘k’ is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. It has dimensions of (L<sup>2</sup> T<sup>-2</sup>), e.g. m<sup>2</sup>/s<sup>2</sup>. ‘ε’ is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate) and has dimensions of k per unit time (L<sup>2</sup> T<sup>-3</sup>), e.g. m<sup>2</sup>/s<sup>3</sup>. The

turbulent kinetic energy equation as modeled has a number of simplifications from the rigorous equation.

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[ \nu + \nu_t / \sigma_k \frac{\partial k}{\partial x_j} \right]$$

$$\tau_{i,j} = -\overline{u_i u_j} = 2\nu_t S_{ij} - \frac{2}{3} k \delta_{ij}$$

The first term on the RHS is the production of ‘k’, the second term (ε) is the specific dissipation per unit mass. The last terms describe the transport of ‘k’ by molecular and turbulent diffusion. Virtually all one and two equation turbulence models solve this equation. To close the ‘k’ equation calculation of ‘ε’ is required and also eddy viscosity ‘ν<sub>t</sub>’ is to be calculate. From dimensional considerations: An exact transport equation for ‘ε’ can be derived by taking the following moment about the Navier- Stokes equations. (The procedure is basically the same as the derivation of rigorous Reynolds Stress Transport equation).

$$2\nu \frac{\partial u_i}{\partial x_j} \frac{\partial}{\partial x_j} [N(u_i)] = 0$$

The standard k-ε model is the default turbulence model in FLUENT. Rather than solving for a length scale it solves a second transport equation for the dissipation rate. This model was derived and tuned for Flows with high Reynolds numbers. This implies that it is suited for flows where the turbulence is nearly iso-tropic and is suited to flows where the energy cascade proceeds in local equilibrium with respect to generation. FLUENT also has the RNG and Realizable k-ε models.

#### i. Steps involved in solving problem

- First create the grid of appropriate dimensions and with appropriate step length to specify the problem domain in Ansys modeler workbench.
- Create geometries like Vertices at appropriate grid points.
- Create lines joining two vertices.
- Create Areas selecting all the lines.
- Create Boundary Mesh around the cylinder.
- Create Face Mesh to rest of the model.
- Give the Boundary Conditions for entire domain.
- Save it and export it to mesh file.
- Read the file in FLUENT and check the mesh and scale the model.

$$\varepsilon = C_d k^{3/2} / l$$

- Enter values for boundary conditions, operating conditions etc.
- Selecting the appropriate solver to solve the problem.
- Solve the problem by initializing from velocity inlet and specifying the number of iterations.
- Solve the problem and note down the results.

$$\nu_t = C_d k^2 / \varepsilon$$

**E. The Advantages of CFD**

Basically, the compelling reasons to use CFD are these three:  
*Insight:* There are many devices and systems that are very difficult to prototype. Often, CFD analysis shows parts of the system or phenomena happening within the system that would not otherwise be visible through any other means. CFD gives a means of visualizing and enhanced understanding of your designs.

*Foresight:* Because CFD is a tool for predicting what will happen under a given set of circumstances, it can answer many ‘what if?’ questions very quickly. Provided the variables it gives you outcomes. In a short time, CFD can predict design performance, and hence number of variants can be tested until optimal result is found. All of this is done before physical prototyping and testing. This foresight made available from CFD helps engineers and designers to design better and faster.

*Efficiency:* Better and faster design or analysis leads to shorter design cycles. Time and money are saved. Products get to market faster. Equipment improvements are built and

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[ \nu + \nu_t / \sigma_k \frac{\partial k}{\partial x_j} \right]$$

$$\frac{\partial \varepsilon}{\partial t} + u_j \frac{\partial \varepsilon}{\partial x_j} = C_{\varepsilon 1} \frac{\varepsilon}{k} \tau_{ij} \frac{\partial u_i}{\partial x_j} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j} \left[ \nu + \nu_t / \sigma_\varepsilon \frac{\partial \varepsilon}{\partial x_j} \right]$$

$$\nu_t = C_\mu k^2 / \varepsilon$$

installed with minimal downtime. CFD is a tool for compressing the design and development cycle.

**i. Applications of CFD**

CFD can predict what will happen, quantitatively, when fluids flow, often with the complications of:

- Simultaneous flow of heat.
- Mass transfer (eg. perspiration, dissolution).
- Phase change (eg. melting, freezing, boiling).
- Chemical reaction (eg. combustion, rusting).
- Mechanical movement (eg. of pistons, fans, rudders).
- Stresses in and displacement of immersed or surrounding solids.

Knowing how fluids will flow, and what will be their quantitative effects on the solids with which they are in contact, assists:

1. Building-services engineers and architects to provide comfortable and safe human environments.
2. Power-plant designers to attain maximum efficiency, and reduce release of pollutants.

Chemical engineers to maximize the yields from their reactors and processing Equipment.

**ii. Limitations of CFD**

CFD-based predictions are never 100%- reliable because of the following reason:

1. The input data may involve too much guess work or imprecision .
2. The available computer power may be too small for high numerical accuracy (in terms of the memory spaces and capabilities).
3. The scientific knowledge base may be inadequate.
4. In terms of the reliability, CFD software differentiates itself with the following aspects.
5. For laminar flows rather than turbulent ones.

6. For single-phase flows rather than multi-phase flows.
7. For chemically-inert rather than chemically-reactive materials.
8. For single chemical reactions rather than multiple ones.
9. For simple fluids rather than those of complex composition.

**V. GEOMETRIC MODELING**

**Geometric Modeling Strategy:** A four column based data relevant to nozzle design has been obtained from ANSYS tutorials and tailored and converted into three columns with Z-coordinate set as zero(as the problem is 2D based) according to Gambit requirements.

**NOZZLE COORDINATES**

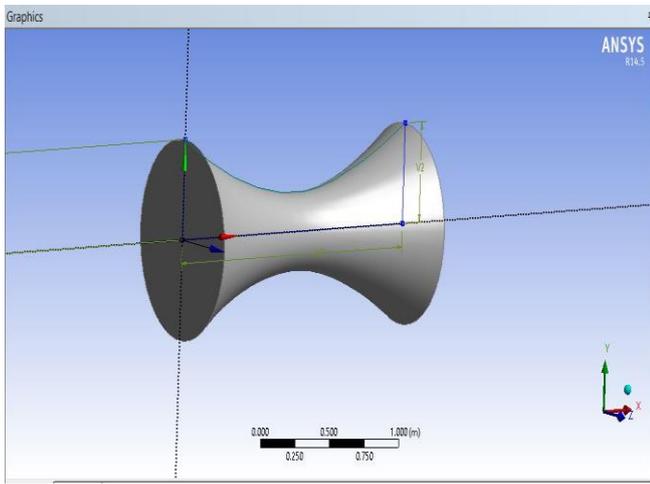
X	R(X)	Z
-0.5	0.333778669	0
-0.45	0.310303908	0
-0.4	0.287681033	0
-0.35	0.266127383	0
-0.3	0.24592425	0
-0.25	0.227431826	0
-0.2	0.211100165	0
-0.15	0.197466123	0
-0.1	0.187120297	0
-0.05	0.180628589	0
0	0.178412203	0
0.05	0.180628589	0
0.1	0.187120297	0
0.15	0.197466123	0
0.2	0.211100165	0
0.25	0.227431826	0
0.3	0.24592425	0
0.35	0.266127383	0
0.4	0.287681033	0
0.45	0.310303908	0
0.5	0.333778669	0
0.5	0.333778669	0
0.5	0	0
0.5	0	0
-0.5	0	0
-0.5	0	0
-0.5	0.333778669	0

The data is saved as a DAT file and is imported into Gambit as ICEM file. The coordinates automatically get dispensed into an upper half profile of the nozzle; no sooner the Accept button is selected. Using Create Edge button, the coordinates of the nozzle are joined, four entities are formed as Inlet, Axis, Wall (comprising of 20 edges) and outlet. Using Create Face button, the edges are converted into a single face.

**NOZZLE DIAGRAM IN ANSYS**



# Designing and Simulating Compressible Flow in a Nozzle

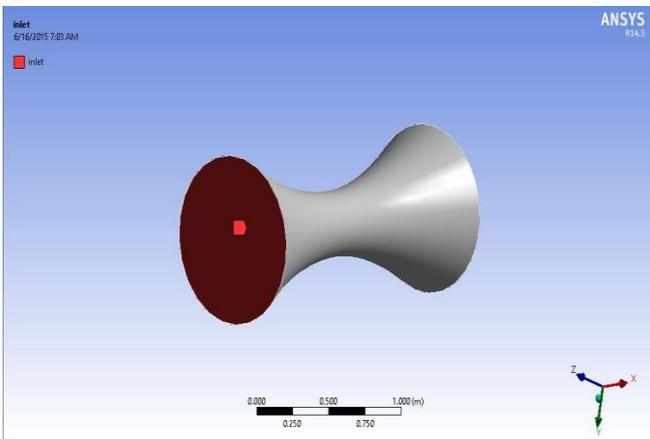


**Boundary Type specification:** It is very essential to specify the boundary types in the Gambit itself, to enable the Fluent understand the boundaries. Left vertical side is set as inlet against Pressure Inlet, right vertical side is assigned Pressure Outlet

## Boundary Types

Edge Position	Name	Type
Left	inlet	PRESSURE INLET
Right	outlet	PRESSURE OUTLET
Top	wall	WALL
Bottom	centerline	AXIS

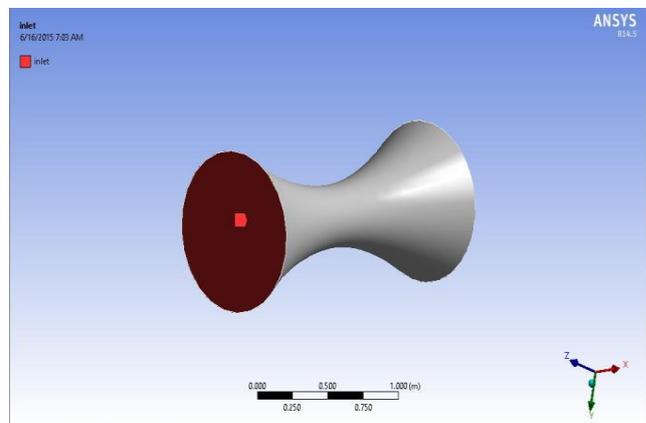
## Inlet of the Nozzle



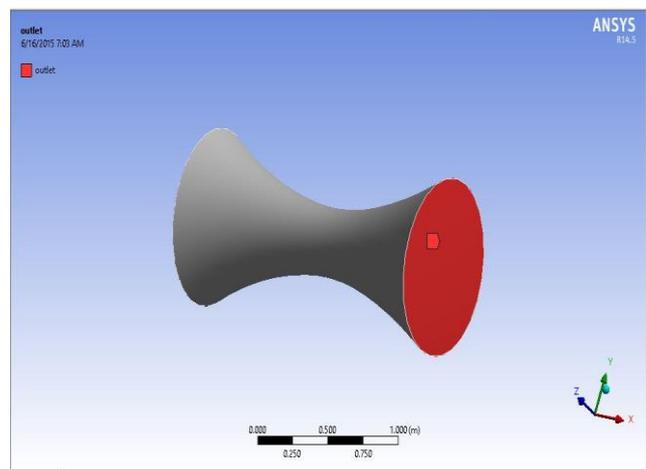
## Problem Set Up In Fluent

Problem Setup	K-Epsilon
1. Solver	Pressure Based
2. Formulation	Implicit
3. Space	Axisymmetric
4. Time Dependency	
5. Velocity Formulation	Steady
6. Gradient Option	
7. Porous Formulation	Absolute
8. Energy Equation	Green-Gauss Cell Based
9. Viscous Model	
10. Viscous Heating	Superficial Velocity ON STD K-Epsilon ON
Material Selection	Properties
Air	Density- Ideal Gas Cp=1006.43 Thermal Conductivity=0.0242 Viscosity=1.7894e-05 Molecular Weight=28.966
Operating Pressure	0 Pascal
Solution Controls	
Equations	Flow Modified Turbulent Viscosity Energy
Under Relaxation Factors	1. Pressure=0.3

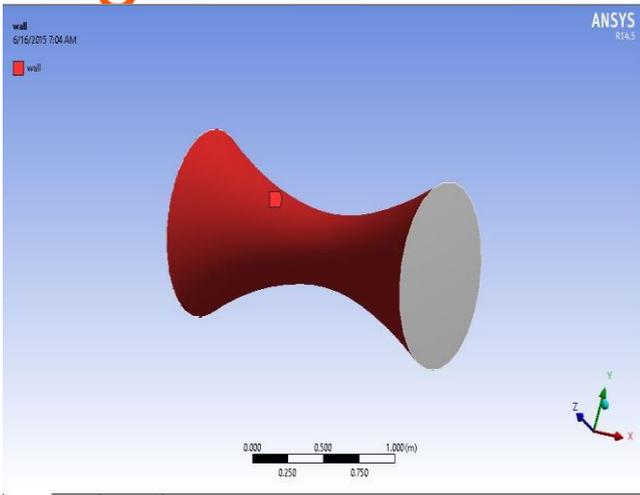
## Showing the Inlet of the Nozzle



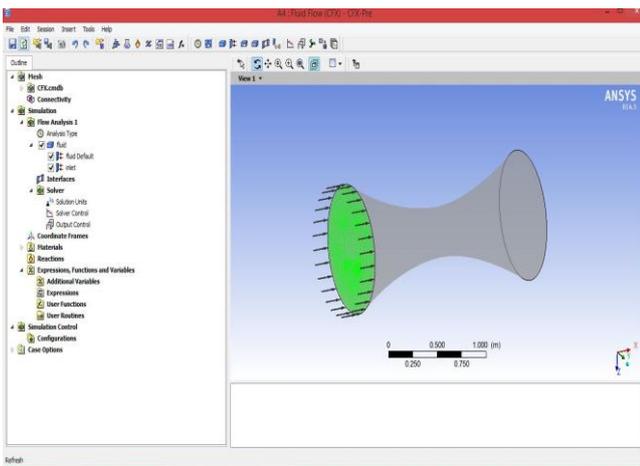
## Out Let of the Nozzle



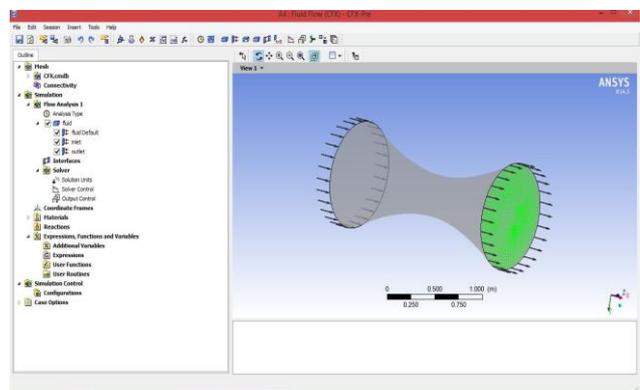
## Wall of the Nozzle



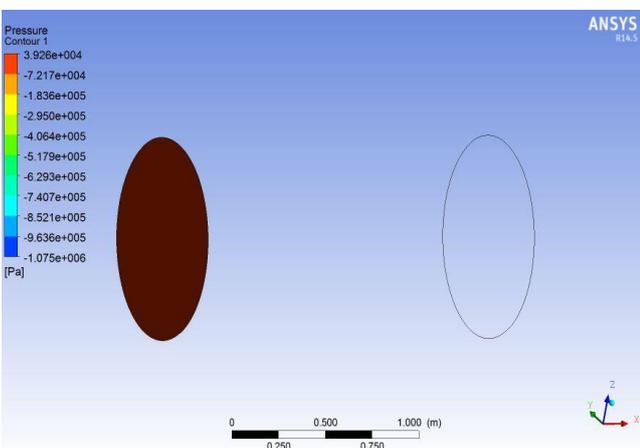
Boundary Condition at the Inlet



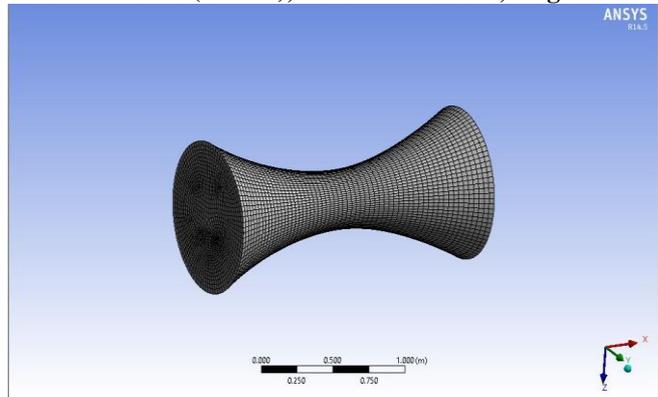
Boundary Condition at the Outlet



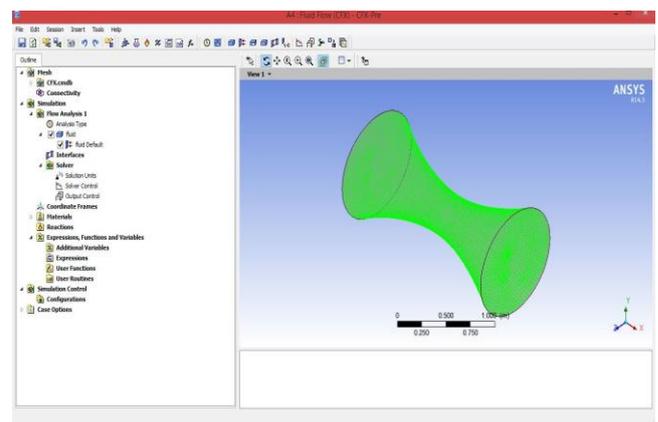
Pressure Inlet Counters



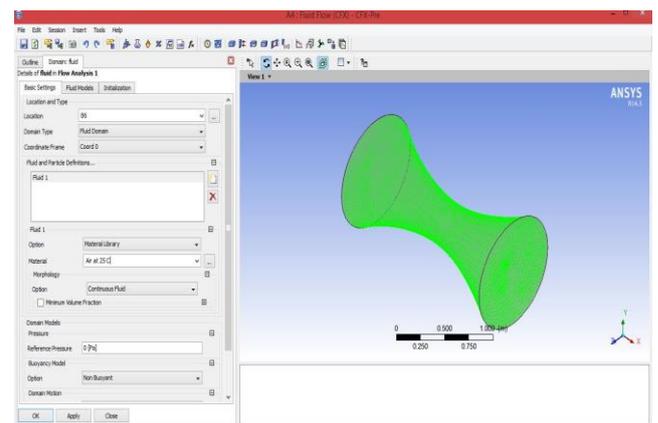
Meshing of the Nozzle



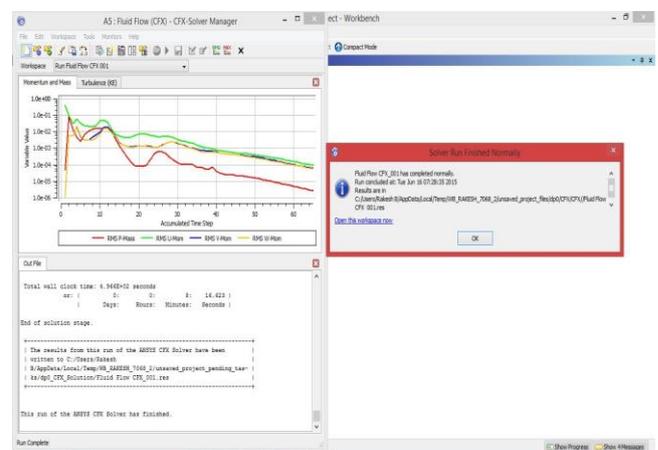
Pre Processor of the Nozzle



Fluid Domain of the Nozzle

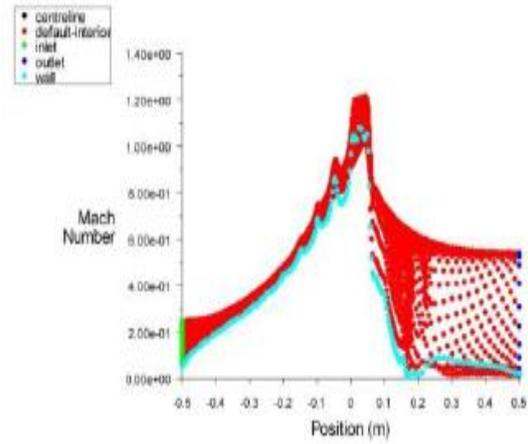
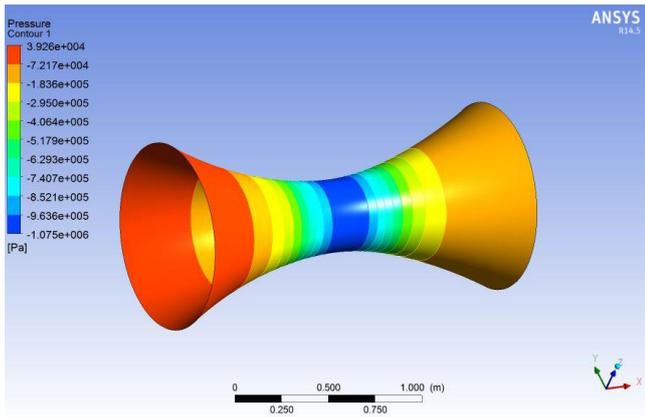


Solution of the Nozzle

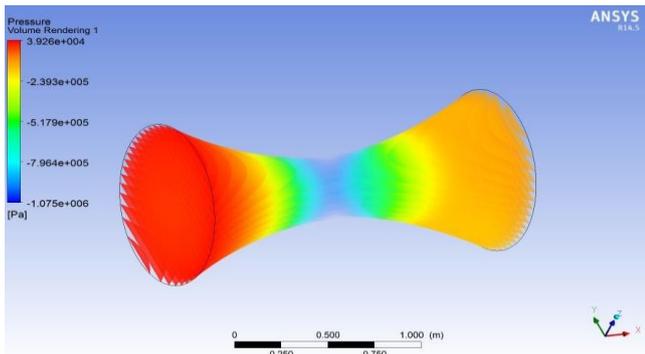


VI. RESULTS

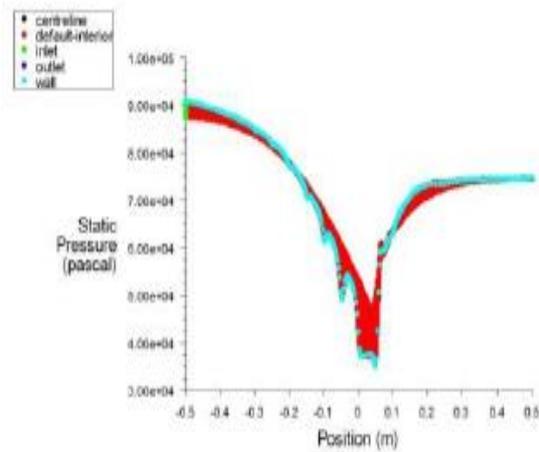
Pressure Counters of the Nozzle



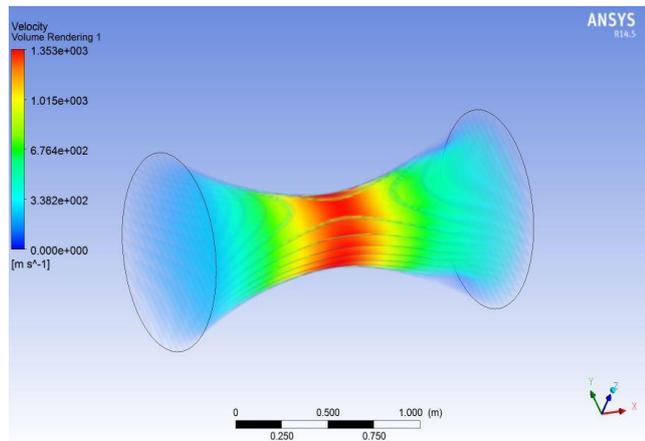
Pressure Volume Rendering of the Nozzle



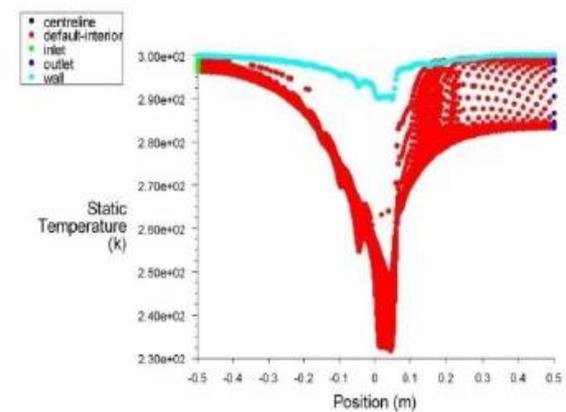
Static Pressure Plot



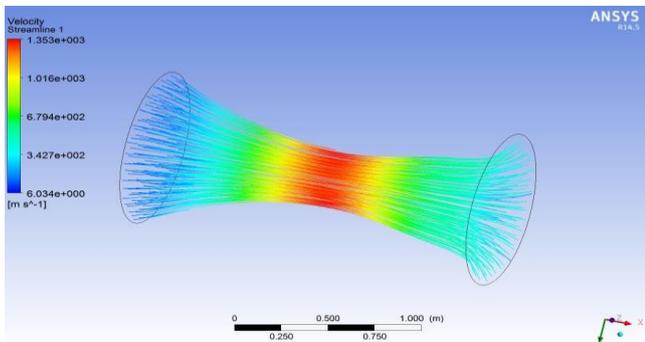
Velocity Volume Rendering of the Nozzle



Static Temperature Plot

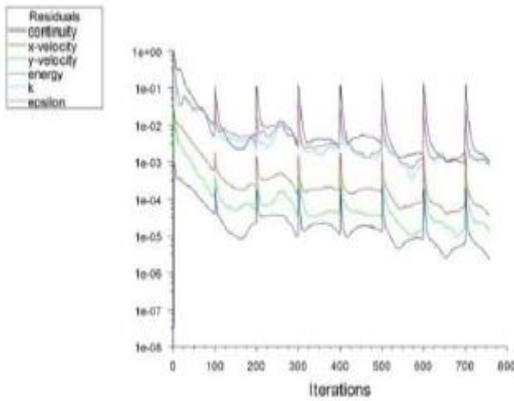


Velocity Streamlines in the Nozzle



Residual Convergent Model

Mach number Plots



## VII. CONCLUSION

The validation of the nozzle under consideration is carried out at an inlet pressure of 101325 pa, outlet pressure of 3738.9 pa, temperature 300 K using inviscid model by comparing Mach No. plot with that of the Quasi 1 D results. It is found that results are in good agreement with each other. Based on the results- plots, contours and vectors obtained from the CFX simulations, following can be concluded:

- A sharp velocity drop is obtained across the shock.
- Flow separation and formation of a vortex after the shock can be attributed to adverse pressure gradient following shocks. Reversal of flow near exit walls is due to the interaction between pressure outlet and vortex. While when the divergence angle is 30 degrees the Mach number at nozzle exit is 3.06 and but an divergence angle 40 degrees it gives the Mach number at nozzle exit is 3.19 and it is lowest at an divergence angle 20 degrees.
- The turbulence intensity is very high for a divergence angle of 20 degrees at exit.
- For maximum velocity we can go with 30 or 40 degrees of divergence angle conical nozzle.
- The efficiency of the nozzle increases as we increase the divergence angle of the nozzle up to certain extent.

## ACKNOWLEDGEMENT

First and foremost, I offer my sincere gratitude to my internal guide **Mr.T.kumaraswamy, Assistant/Associate Professor** of Aeronautical Engineering who has supported me throughout this project with her/his patience and valuable suggestions. I would like to express my gratitude to **Dr.M. Satyanaraya Gupta** Head of the Department Aeronautical Engineering for his support and valuable suggestions during the dissertation work. I am also grateful to the Principal **Dr. P Bhaskar Reddy** for providing me with all the resources in the college to make my project a success. I thank him for his valuable suggestions at the time of seminars which encouraged me to give my best in the project. I would also like to thank all the supporting staff of the Dept. of ANE and all other departments who have been helpful directly or indirectly in making the project a success. I am extremely grateful to my parents for their blessings and prayers for my completion of project that gave me strength to do my project.

## REFERENCES

- [1] P Manna, D Chakraborty “Numerical Simulation of Transverse H2 Combustion in Supersonic Airstream in a Constant Area Duct”, Vol. 86, November 2005, computational combustion Dynamics Division of Defense Research and Development Laboratory, Hyderabad.
- [2] B.E. Milton and K. Pianthong, “Pulsed, supersonic fuel jets—A review of their characteristics and potential for fuel injection”, International Journal of Heat and Fluid Flow 26 (2005) 656–671, Australia.
- [3] Shigeru Aso, ArifNur Hakim, Shingo Miyamoto, Kei Inoue and Yasuhiro Tani “ Fundamental study of supersonic combustion in pure air flow with use of shock tunnel” Department of Aeronautics and Astronautics, Kyushu University, Japan , ActaAstronautica 57 (2005) 384 – 389.
- [4] Chadwick C. Rasmussen, Sulabh K. Dhanuka, and James F. Driscoll, “Visualization of flame holding mechanisms in a supersonic combustor using PLIF”, Proceedings of the Combustion Institute 31 (2007) 2505–2512, USA.
- [5] P.K. Tretyakov “the problems of combustion at supersonic flow” west-east high speed flow field conference 19-22, November 2007 Moscow, Russia.
- [6] Zheng Chen, Xiao Qin, YiguangJu \*, Zhenwei Zhao, Marcos Chaos, Frederick L. Dryer, “High temperature ignition and combustion enhancement by dimethyl ether addition to methane–air mixtures”, Proceedings of the Combustion Institute 31 (2007) 1215–1222, USA.
- [7] DoyoungByun and SeungWookBaek, “Numerical investigation of combustion with non-gray thermal radiation and soot formation effect in a liquid rocket engine”, International Journal of Heat and Mass Transfer 50 (2007) 412–422, Korea.
- [8] Wookyung Kim, Hyungrok Do, M. Godfrey Mungal and Mark A. Cappelli, “Optimal discharge placement in plasma-assisted combustion of a methane jet in cross flow”, Combustion and Flame 153 (2008) 603–615, USA.
- [9] Peter Gerlinger, Peter Stoll 1, Markus Kindler , Fernando Schneider c, Manfred Aigner “Numerical investigation of mixing and combustion enhancement in supersonic combustors by strut induced streamwisevorticity”, Aerospace Science and Technology 12 (2008) 159–168, Germany
- [10] K. Kumaran, V. Babu “Investigation of the effect of chemistry models on the numerical predictions of the supersonic combustion of hydrogen”, Department of Mechanical Engineering, Indian Institute of Technology, Madras, India, Combustion and Flame 156 (2009) 826–841.
- [11] Kenji Miki, Joey Schulz, Suresh Menon “Large-eddy simulation of equilibrium plasma-assisted combustion in supersonic flow”, Proceedings of the Combustion Institute 32 (2009) 2413–2420, Atlanta, GA 30332-0150, USA.
- [12] J.X. Wen\*, B.P. Xu and V.H.Y. Tam, “Numerical study on spontaneous ignition of pressurized hydrogen release through a length of tube”, Combustion and Flame 2009, UK.