# Modelling and Computational Fluid Dynamic Analysis on Jet Nozzle

[1]Shaik Khaja Hussain, [2]B V Amarnath Reddy, [3]A V Hari Babu

[1]Research Scholar, [2]Assistant Professor, [3]HOD
Mechanical Engineering Department
AVR & SVR college of Engineering and technology nandyal

***Abstract*: Supersonic flows associated with missiles, aircraft, missile engine intake and rocket nozzles are often steady. The shape of the nozzle geometry is increasingly attractive in heating, ventilation and air conditioning applications. The objective of the present work is to simulate and understand Supersonic flows with single jet Flow at various Mach numbers. The purpose is to precisely understand the fluid dynamics and variation of flow properties such as velocity, pressure and turbulence in supersonic flow regime for various Mach numbers pressure ratio and dimensionless spacing single jet nozzle. The jet nozzle can take three different materials like Titanium, Tungsten and Nickel. It can observe the material is to be in the supersonic flow in the nozzle. The Mach number at the nozzle exit is observed to be less in comparison with designed value. This is due to the viscosity and turbulence in fluid near the wall of the duct. The Mach number decreases due to shock wave and reversible flow. A single jet nozzle is designed in solidworks software and CFD analysis is carried out in solid works flow simulation.**

***Keywords*: Supersonic nozzle, CFD Analysis solid works.**

## 1. INTRODUCTION

The development of the gas turbine engine as an aircraft power plant has been so rapid that it is difficult to appreciate that prior to the 1950s very few people had heard of this method of aircraft propulsion. The possibility of using a reaction jet had interested aircraft designers for a long time, but initially the low speeds of early aircraft and the unsuitably of a piston engine for producing the large high velocity airflow necessary for the ëjetí presented many obstacles.

A French engineer, RenÈ Lorin, patented a jet propulsion engine in 1913, but this was an athodyd and was at that period impossible to manufacture or use, since suitable heat resisting materials had not then been developed and, in the second place, jet propulsion would have been extremely inefficient at the low speeds of the aircraft of those days. However, today the modern ramjet is very similar to Lorin's conception.

In 1930 Frank Whittle was granted his first patent for using a gas turbine to produce a propulsive jet, but it was eleven years before his engine completed its first flight. The Whittle engine formed the basis of the modern gas turbine engine, and from it was developed the Rolls-Royce Welland, Derwent, Nene and Dart engines.

The Derwent and Nene turbo-jet engines had world-wide military applications; the Dart turbo-propeller engine became world famous as the power plant for the Vickers Viscount aircraft. Although other aircraft may be fitted; with later engines termed twin-spool, triple-spool, by-pass, ducted fan, unducted fan and prop fan, these are inevitable developments of Whittle's early engine.

The jet engine although appearing so different from the piston engine-propeller combination, applies the same basic principles to effect propulsion both propel their aircraft solely by thrusting a large weight of air backwards.

The test is virtualized at different Mach numbers, where the flow conditions are derived. The virtualization is one of the major developments in the field of research, which revolutionized Aerospace engineering, along with all other branches. The computational techniques are used widely for getting better results, close to experimental techniques.

The flow through a converging-diverging nozzle is used for modeling the compressible flow through computational fluid dynamics. Accurate shock predication is a challenge to the CFD fraternity. But in reality, multi dimensionality and viscous effects like wall boundary layer and flow separation drastically alter the flow in a CD nozzle. The prediction of such flows also presents a great challenge to any CFD code.

## 2. LITERATURE

Jet engine nozzle is designed for attaining speeds that are greater than speed of sound. The design of this nozzle came from the area-velocity relation (dA/dV) =-(A/V)(1- M^2) M is the Mach number ( which means ratio of local speed of flow to the local speed of sound) A is area and V is velocity The following information can be derived from the area-velocity relation –

1. For incompressible flow limit, i.e. for M tends to zero, AV = constant. This is the famous volume conservation equation or continuity equation for incompressible flow.

2. For M < 1, a decrease in area results in increase of velocity and vice versa. Therefore, the velocity increases in a convergent duct and decreases in a Divergent duct. This result for compressible subsonic flows is the same as that for incompressible flow.

3. For M > 1, an increase in area results in increases of velocity and vice versa, i.e. the velocity increases in a divergent duct and decreases in a convergent duct. This is directly opposite to the behavior of subsonic flow in divergent and convergent ducts.

4. For M = 1, dA/A = 0, which implies that the location where the Mach number is unity, the area of the passage is either minimum or maximum. We can easily show that the minimum in area is the only physically realistic solution.

One important point is that to attain supersonic speeds we have to maintain favorable pressure ratios across the nozzle.

One example is attain just sonic speeds at the throat, pressure ratio to maintained is (P throat / P inlet) = 0.528.

Table 1: mach number regime

`

| Regime | Subsonic | Transonic | Sonic | Supersonic | Hypersonic | High-hypersonic |
|---|---|---|---|---|---|---|
| Mach | < 1.0 | 0.8-1.2 | 1.0 | 1.0-5.0 | 5.0-10.0 | >10.0 |

rom table.1 at transonic speeds, the flow field around the object includes both sub- and supersonic parts. The transonic period begins when first zones of M > 1 flow appear around the object. In case of an airfoil (such as an aircraft's wing), this typically happens above the wing.

Supersonic flow can decelerate back to subsonic only in a normal shock; this typically happens before the trailing edge (Fig.a).

As the speed increases, the zone of M > 1 flow increases towards both leading and trailing edges. As M=1 is reached and passed, the normal shock reaches the trailing edge and becomes a weak oblique shock: the flow decelerates over the shock, but remains supersonic. A normal shock is created ahead of the object, and the only subsonic zone in the flow field is a small area around the objects leading edge (Fig.b).

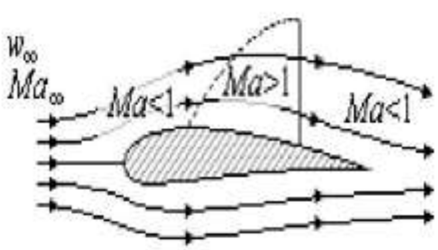



**Figure (a)**

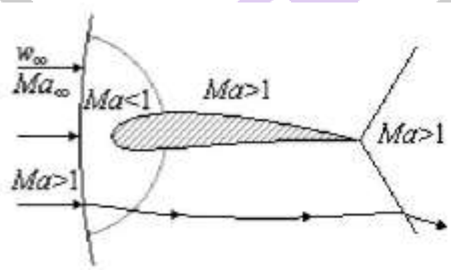



**Figure (b)**

The governing continuity, momentum, and energy equations for this quasi one-dimensional, steady, isentropic flow can be expressed, respectively as

Continuity:

$$\rho_1 A_1 V_1 \quad \rho_2 A_2 V \qquad (1)$$

Momentum:

$$p_1 A_1 \quad \rho_1 V_1^2 A_1 \quad \int_{A_1}^{A_2} p dA \quad p_2 A_2 \quad \rho_2 V_2^2 A_2 \qquad (2)$$

Energy:

$$h_1 + \frac{V_1^2}{2} = h_2 + \frac{V_2^2}{2} \qquad (3)$$

Where subscripts 1 and 2 denote different locations along the nozzle. In addition, we have the perfect gas equation of state,

$$p = \rho R T \qquad (4)$$

As well as the relation for a calorically perfect gas,

$$h = c_p T \quad (5)$$

Equations (1) and (5) can be solved analytically for the flow through the nozzle

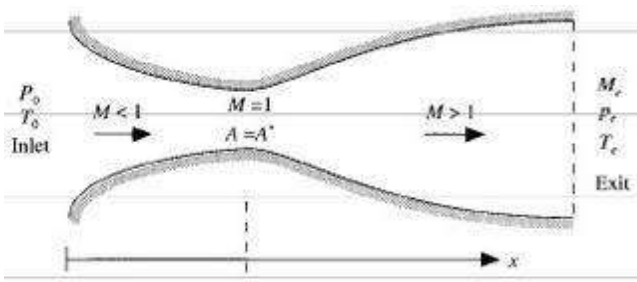



**Figure (c)**

**2.1 Assumed Model**

Steady, quasi-one-dimensional. There are gradual variations in the geometry, so that the flow near the nozzle walls is not strictly along the x-direction. However, the flow angularity is very small. The variation in properties can be calculated assuming that the properties are constant in each cross-section. The cross-section area, A, is a function of x alone. Thus, all properties are functions of x alone.

A= A(x); u = u(x); T=T(x), p= p(x) etc.

$$\rho u A = \dot{m} \quad (6)$$

the mass flow rate, is constant.

$$\frac{dA}{A} + \frac{d\rho}{\rho} + \frac{du}{u} = 0 \ldots\ldots\ldots\ldots\ldots (7)$$

Momentum: (no friction; differential form of the Euler equation)

$$udu = -dp/\rho \quad (8)$$

Hence

$$\frac{du}{u} = \frac{dp}{\rho u^2} \ldots\ldots\ldots\ldots\ldots\ldots (9)$$

Using (8) in (10),

$$\frac{dA}{A} = \frac{dp}{\rho u^2} \quad \frac{d\rho}{\rho} = \frac{dp}{\rho u^2}\left(1 \quad \frac{u^2}{\left(\frac{dp}{d\rho}\right)}\right) \quad (10)$$

Isentropic process

$$\left(\frac{dp}{d\rho}\right) = a \quad (11)$$

Thus,

$$\frac{dA}{A} = \frac{dp}{\rho u^2}(1 - M^2) \quad (12)$$

Also,

$$\frac{\;}{A} = -\frac{\;}{u}(1 - M^2) \quad (13)$$

Case 1: M<1

dA, dp have the same sign. Thus, as A increases, p increases. dA, du have opposite signs. Thus as A increases, u decreases.

**Diverging duct in subsonic flow** : pressure increases, speed decreases.
**Converging duct in subsonic flow:** pressure decreases, speed increases.

Case 2: M>1

dA, dp have opposite signs. Thus as A increases, p decreases. dA, du have the same sign. Thus as A increases, u increases.

**Diverging duct in supersonic flow:** pressure decreases, speed increases.

Case 3: M = 1

dA/dx is 0. Thus we have either a maximum or minimum of area.

The maximum area case is not of much interest, since there is no way to reach Mach 1 at this point, with flow from either direction. So the case of interest is where the area becomes a minimum: a "throat".

From mass conservation, $\rho u A = \rho^* u^* A^*$ where the * denotes conditions at Mach 1 So,

$$\frac{A}{A^*} = \frac{\rho^*}{\rho^*}\frac{u^*}{u} \qquad (14)$$

$$- = \frac{}{\rho_0}\frac{}{\rho} = \left(\frac{2}{\gamma+1}\right)^{\frac{-1}{\gamma-1}}\left(1 + \frac{}{2}M^2\right)^{\frac{-1}{\gamma-1}} \qquad (15)$$

$$\frac{u^*}{} = \frac{}{a}\frac{}{u} \qquad (16)$$

$$- = \frac{}{aM} = \frac{}{a_0}\frac{}{aM} = \left(\frac{2}{\gamma+1}\right)^{\frac{\gamma}{\gamma-1}}\left(1 + \frac{}{2}M^2\right)^{\frac{\gamma}{\gamma-1}}\left(\frac{1}{M}\right) \qquad (17)$$

$$\frac{}{p_0} = \left(1 + \frac{}{2}M^2\right)^{\gamma/(1-\gamma)} \qquad (18)$$

$$\frac{}{\rho_0} = \left(1 + \frac{}{2}M^2\right)^{\frac{-1}{\gamma-1}} \qquad (19)$$

$$\frac{T}{T_0} = \left(1 + \frac{}{2}M^2\right)^{-1} \qquad (20)$$

Substitute into A/A*:

$$\frac{}{A^*} = \frac{}{M}\left[\frac{2}{\gamma+1}\left(1 + \frac{}{2}M^2\right)\right]^{(\gamma+1)/(1-\gamma)} \qquad (21)$$

Thus, for a given isentropic flow, i.e., a flow with mass flow rate, stagnation temperature and stagnation pressure all fixed, there are two solutions for a given value of A/A*: One solution is subsonic, the other is supersonic.

### 2.2 Mass Flow Rate Through a Nozzle

For given stagnation conditions $\rho u A \rho^*_o u^*_o A^*$ are fixed.

$$\rho^* u^* \quad \dot{m} \quad maximum\left(\frac{\dot{m}}{A}\right) \qquad (22)$$

For a given throat area, stagnation pressure and stagnation temperature, the maximum mass flow rate is the value where the Mach number at the throat reaches 1.0. This is called the "choked mass flow rate." To increase the mass flow rate, we have to increase the stagnation pressure, decrease the stagnation temperature, or increase the throat area.

$$\frac{\dot{m}}{A} = \rho u = \frac{pMa}{RT} = \frac{pM\sqrt{\gamma RT}}{RT}, \text{ or}$$

$$\frac{\dot{m}}{A} = \left(\frac{p}{p}\right)\left(\frac{T_o}{T}\right)\left(\sqrt{\frac{T}{T_o}}\right)M\left(\sqrt{\frac{\gamma}{R}}\right)\left(\frac{p_o}{\sqrt{T_o}}\right)$$

$$\frac{\dot{m}}{A} = \left(\sqrt{\frac{\gamma}{R}}\right)\left(\frac{p_o}{\sqrt{T_o}}\right)\frac{M}{\left(1-\left(\frac{\gamma-1}{2}\right)M\right)^{\frac{(\gamma+1)}{2(\gamma-1)}}} \qquad (23)$$

$$\left(\frac{\dot{m}}{A}\right)_{max} = \frac{p_0}{\sqrt{T_0}}\left(\frac{2}{\gamma+1}\right)^{(\gamma+1)/2(\gamma-1)}\sqrt{\frac{\gamma}{R}} \qquad (24)$$

For M=1, R=286.7 J/Kg K and $\gamma$ = 1.4 for air,

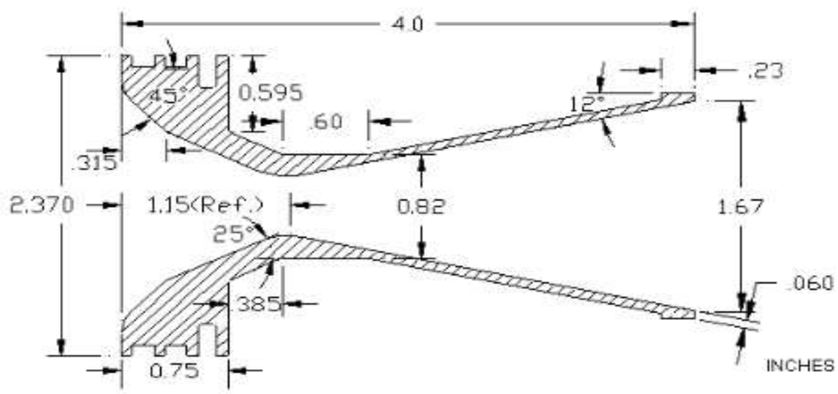
$$\left(\frac{\dot{m}}{A}\right)_{max} = 0.04044\,\frac{p_0}{\sqrt{T_0}}\,\frac{kg/s}{m^2} \qquad (25)$$

Where $p$ is in $N/m^2$, $T$ in Kelvin and $A$ in $m^2$.

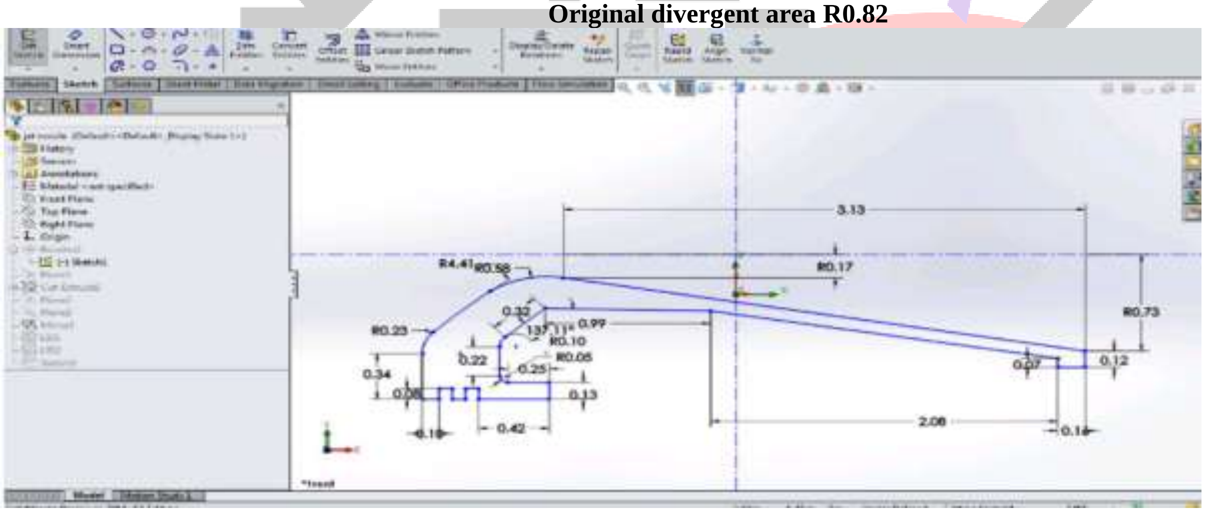
**3. Methodology and Implementation**

**3.1 Modeling the Nozzle**

Supersonic nozzles are generally specified in terms of the cross sectional area of final uniform flow A and the final mach number M. The nozzle-throat area is obtained by the 1D flow equation; the shortest nozzles that may be designed by the method of reported are those without a straight-walled section. The straightening part immediately follows with the expanding part. The purpose of method of characteristics is to illustrate the design of a supersonic nozzle by the method of computation we have developed the model in such a way that it is assumed to be in the mid-section of the nozzle.




**Original divergent area R0.82**




Modified part: **Divergent area has changed to R0.73**

By revolving the above sketch we get the model of jet nozzle.

**Flow simulation of jet nozzle:**

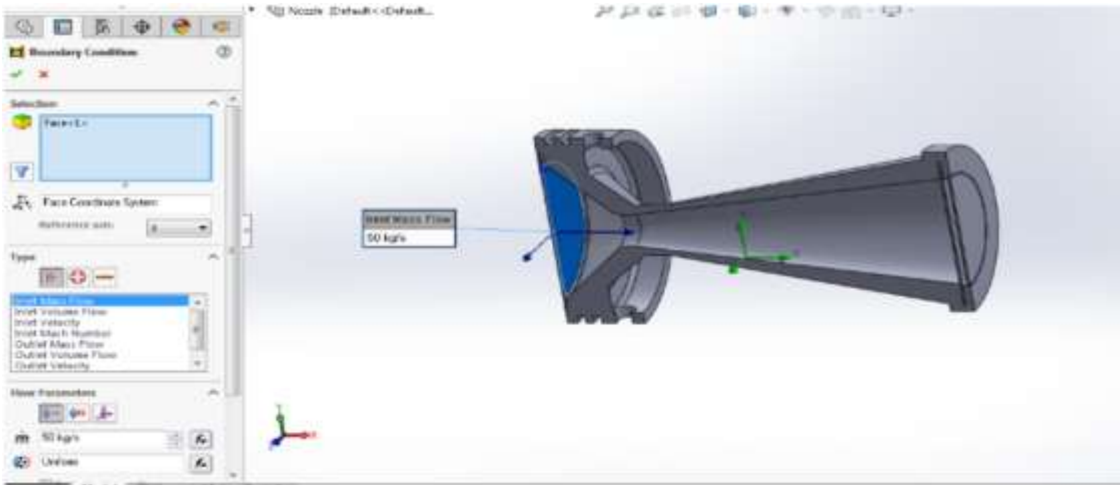
**Applying boundary conditions on jet nozzle in solid works flow simulation**

**Setting the units**

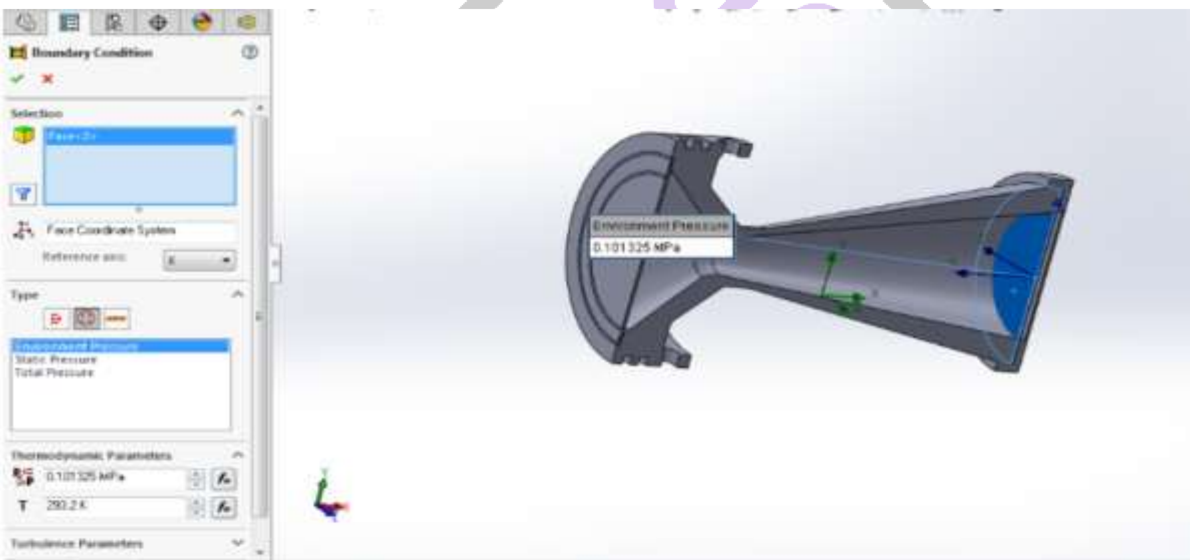
1. Using default internal analysis and checking the heat conduction in solids box

2. 2.selectfluidair(gases).

3. 3. Results and geometric resolutions.




**Mass flow inlet 50 kg/sec**
**At out let atmospheric pressure is taken**




Solving the flow simulation the results will be shown at nozzle.

### 3.2 Boundary Conditions Used

1. Mass flow inlet

2. Outlet atmospheric temperature

### 3.3 Solving

Fluent analysis is carried out for nozzle at different Mach numbers and at different Nozzle pressure ratios. The steady axi symmetric implicit formulation with coupled solver with the mass flow of the burning solid propellant was modeled using a user-defined function.

### 3.4 Solid works Flow Simulation:

Solid Works Flow Simulation 2010 is a fluid flow analysis add-in package that is available for Solid Works in order to obtain solutions to the full Navier-Stokes equations that govern the motion of fluids. Other packages that can be added to SolidWorks include SolidWorks Motion and SolidWorks Simulation. A fluid flow analysis using Flow Simulation involves a number of basic steps that are shown in the following flowchart in figure.

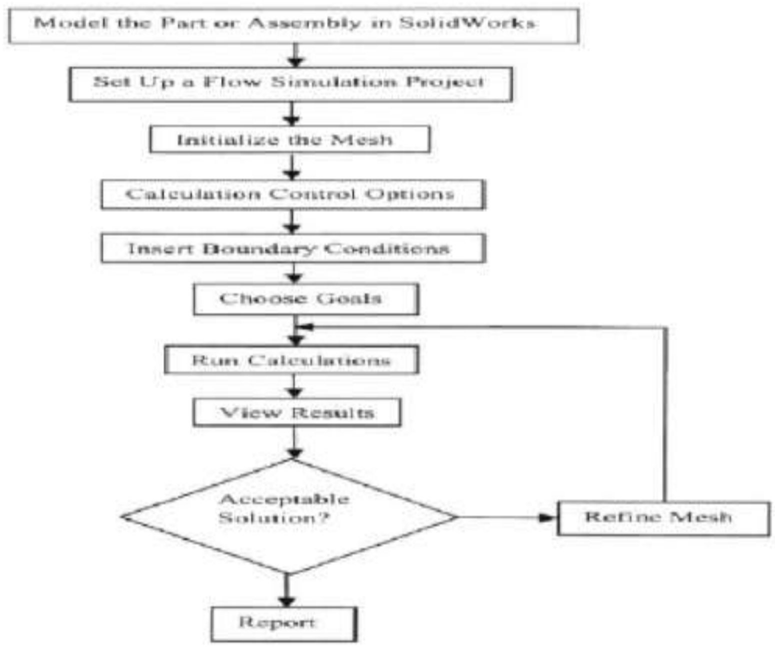



Figure: Flowchart for fluid flow analysis using Solidworks Flow Simulation **Setting up a Solidworks.**

## 4. RESULTS

### 4.1 NOZZLE Analysis for Determination of Pressure, Velocity, and mach number

Fluent analysis is carried out at different Mach numbers and the values are tabulated at Different Mach Numbers. The pressure contours as shown gives us the variation of static pressure across the nozzle. The pressure decreases from inlet to outlet of the nozzle, during which pressure energy is converted into kinetic energy. In converging section the velocity increases and mach number reaches at the throat and it increases in the divergent section until the exit of the nozzle at the expensive of pressure and temperature. We can also use the variation of static pressure along the nozzle. The temperature at the inlet is maximum because the combustion gases are high temperature and it decease along the nozzle due to expansion.

In these are two material can be taken and it can be observes at different pressures, velocity and mach number of the nozzle at the modeling. in these tungsten and titanium can be taken and it performance can be evaluate.

The variation of static pressure, temperature and mach number are shown in figures below. In contour nozzle, the loss of thrust component is less when compared to conical nozzle and this can be seen in mach number contour that mach number is maximum at axis of exit section. The velocity is maximum at the axis and it decreases as we move towards wall. The variation of static temperature is minimum at the axis of exit section than the wall.

**TUNGSTEN**

| | Pressure (bar) | Velocity (m/s) | Mach number |
|---|---|---|---|
| **Max** | **2.2782** | **694.633** | **3.67** |
| **Min** | **0.1672** | **0** | **0** |

Table no.1

**TITANIUM**

| | Pressure (bar) | Velocity (m/s) | Mach number |
|---|---|---|---|
| **Max** | 2.389 | 681.359 | 5.73 |
| **Min** | 0.1325 | 0 | 0 |

Table no.2

The following figures illustrate Contours of Pressure, Velocity Magnitude and Mach numbers.

**TUNGSTEN**

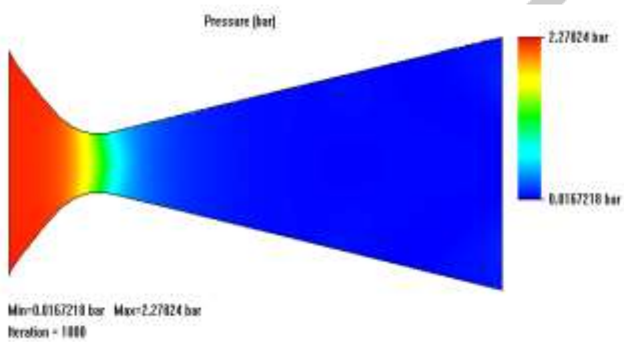



**Fig: 1 Variation Contour of Pressure**

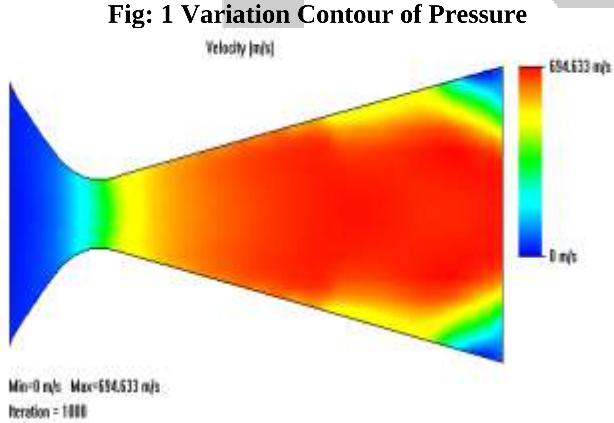



**Fig: 2 Variation Contour of Velocity**

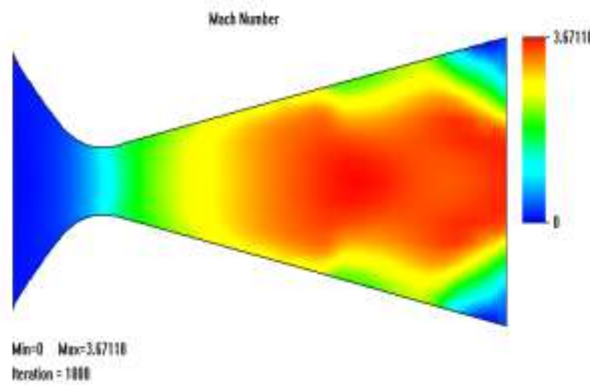



**Fig : 3Contour of Mach number**

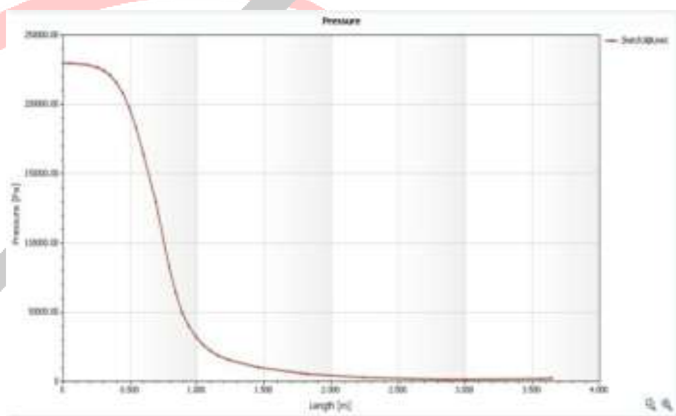



**Fig: 4 total Static Temperature performance**

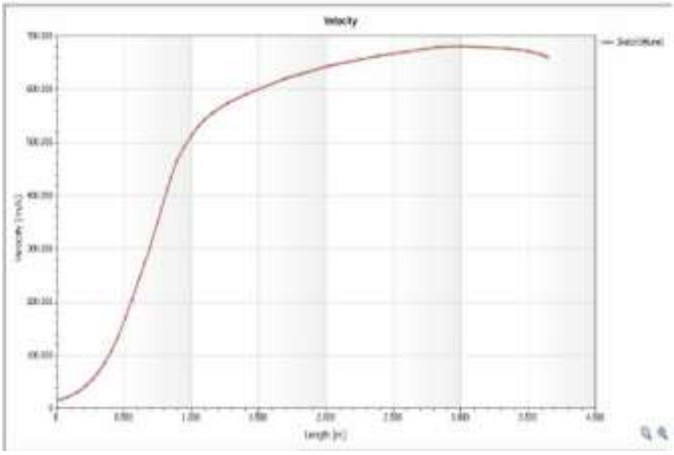



**Fig: 5 total velocity performance**

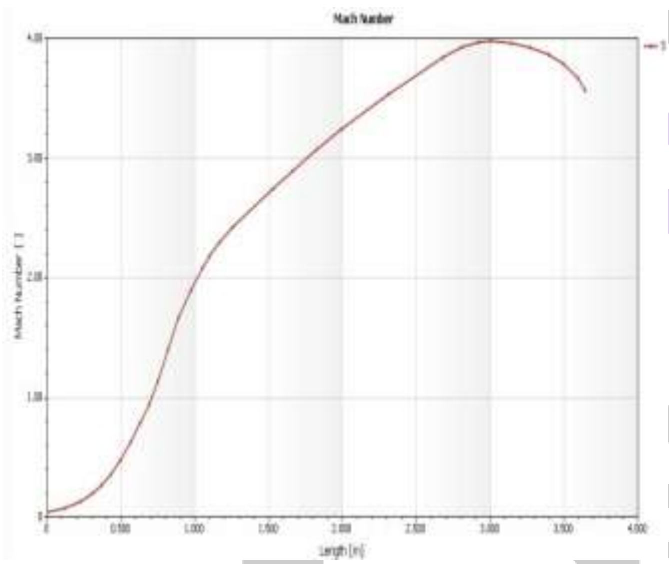



**Fig: Mach number performance**

**Titanium**

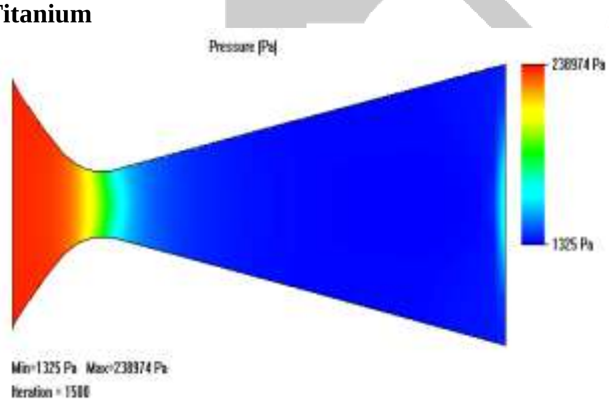



**Fig: 1 Variation Contour of Pressure**

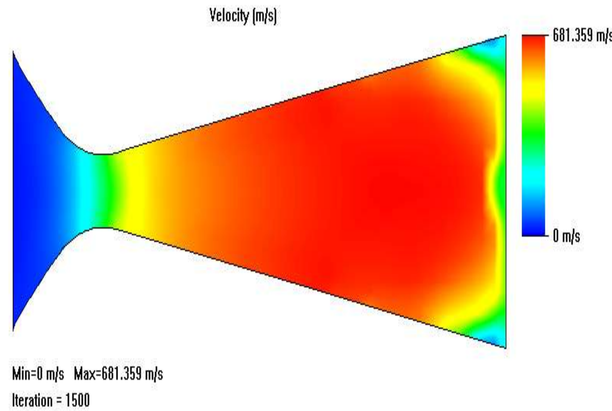



**Fig: 2 Variation Contour of Velocity**

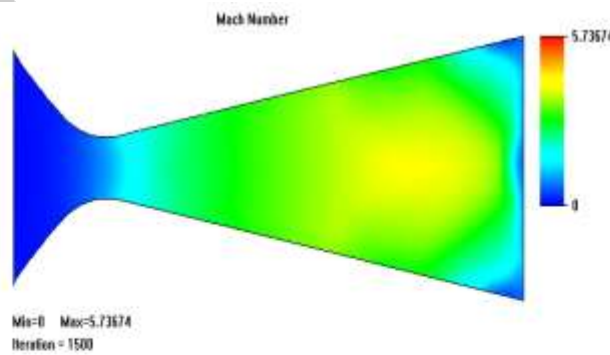



**Fig: 3Mach number**

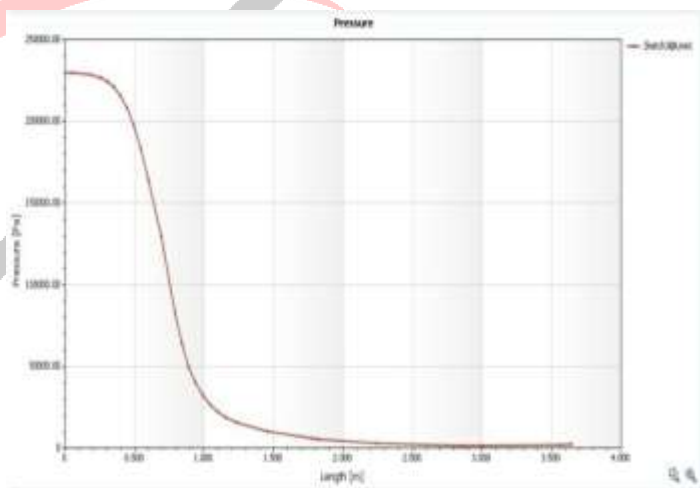



**Fig: 4 total Static Temperature performance**

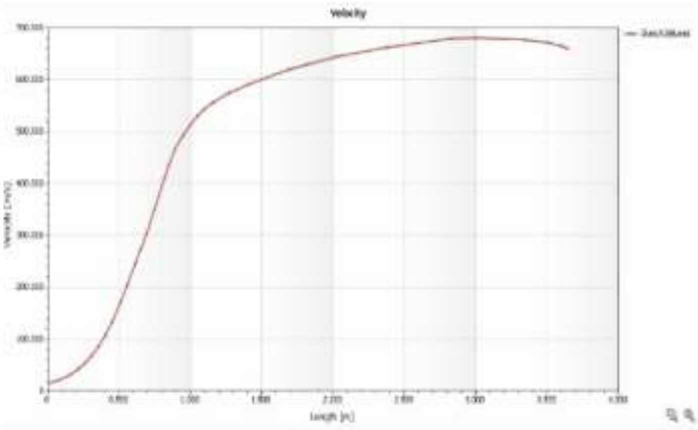



**Fig: 5 total velocity performance**

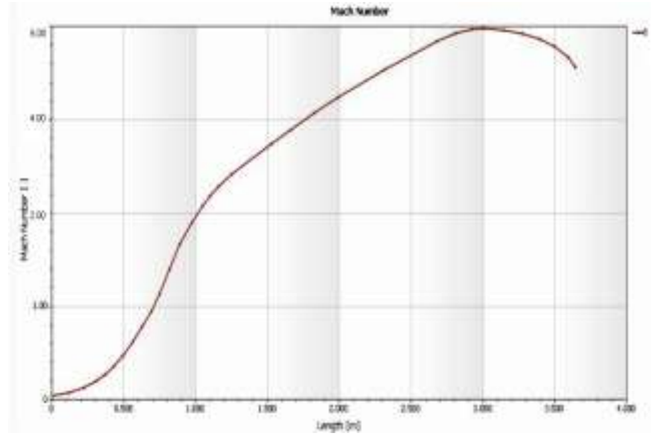



**Fig: 6 Mach number performances**

## 5. Conclusion

Computer aided solutions are developed using Fluent Analysis. Solutions are evaluated at different Mach numbers corresponding Mass flow rates, Maximum Velocity, Maximum Pressure are determined. Variation in static pressure increases with Mach number less than 5 and was found to be identical from the inlet throat to the exit for Mach number values higher than 5.With the corresponding increase of Mach number corresponding velocity also increases. Maximum forces alternately increase and decrease with respect to Mach number.

Boundary conditions were taken inlet mass flow rate has given 50kg/sec and outlet flow is atmospheric pressure, Divergent area has changed from R0.82 to R0.73. Highest mach number is obtained at original divergent area. Pressure is decreased at convergent region due to sudden expansion of flue gasses. If we vary the throat area and convergent area it may increase the performance of jet nozzle.

Fluent is utilized to simulate the transient gas flow by a coupled explicit solver and it gives a 2-D result. An overall first order and second order scheme is employed spatially and temporally. Simulated Pressure histories, Temperature histories and Mach number distributions agree well with the corresponding reported static and pilot pressure measurements. Comparison of the Pressure ratio, Density ratio, Temperature ratio and Mach numbers are done between the analytical and fluent output.

## 6. References:

[1] A.A.Khan and T.R.Shem bharkar, 2008, Viscous flow analysis in a convergent – Divergent nozzle, International Journal of Computational Engineering Research, IJCERonline, India, Volume 3, No. 5, pp. 5-15.

[2] Du. H, Liu J, Tang J, 2008, A CFD investigation on the nozzle of orifices distributing in different space layers, SAE International, SAE World Congress and Exhibition, USA, 2008-01-0948.

[3] DR. Keerthana and G. Jamuna Rani, 2012, Flow analysis of Annular Diffusers, International Journal of Engineering Research and Application, India, Volume 2, No. 3, pp. 2348-2351.

[4] J Hussain AKMF, Ramjee V, 2010, Effects of the axisymmetric contraction shape on incompressible turbulent flow, Journals of Fluid Engineering, ASME, USA,Volume 98, No. 1, pp. 56-68.

[5] Kazuhiro Nakahashi, 1989, Navier-Stokes Computations of two and three dimensional cascade flow fields, Aerospace Research Central, AIAA, Japan, Volume 5, No. 3, pp. 320-326.

[6] K.M.Pandey, Virendra Kumar, 2010, CFD Analysis of Twin Jet Flow at Mach 1.74 with Fluent Software, International Journal of Environmental Science and Development, India, Volume 1, No. 5, pp. 423-427.

[7] Manoj Kumar Gopaliya, 2014, Performance Improvement of S-shaped Diffusers Using Momentum Imparting Technique, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE), India, Volume 11, No.3 , pp. 23-31.

[8] Mohan Kumar G, 2013, Design and Optimization of De Lavel Nozzle to Prevent Shock Induced Flow Separation, Advances in Aerospace Science and Applications, India, Volume 3, No.2, pp. 157-165.

[9] Ms. B. Krishna Prafulla, Dr. V. Chitti Babu, Sri P. Govinda Rao, 2013, CFD Analysis of Convergent Divergent Supersonic Nozzle.