DESIGN AND FLOW ANALYSIS OF CONVERGENT DIVERGENT NOZZLE USING CFD

A dissertation submitted in the partial fulfilment of
the requirements for the award of the degree of

BACHELOR OF TECHNOLOGY

In

MECHANICAL ENGINEERING

By

P SAI TEJA	**315126520177**
B.S.S.PRUDHVI	**315126520193**
P.RAMESH APPALARAJU	**315126520179**
T.V.R.S.S.N.PRUDHVI	**315126520214**
P.SANDEEP	**315126520167**

Under the Guidance of

Mr. S. RAMANJANEYULU

.

Assistant Professor

PRAGNANAM BRAHMA
ANITS

DEPARTMENT OF MECHANICAL ENGINEERING
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES (A)
Sangivalasa, Bheemunipatnam, Visakhapatnam (dist.),
Andhra Pradesh, India. 531162
2015 - 2019

DEPARTMENT OF MECHANICAL ENGINEERING
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES (A)

PRAGNANAM BRAHMA
ANITS

CERTIFICATE

This is to certify that the thesis entitled **"DESIGN AND FLOW ANALYSIS OF CONVERGENT DIVERGENT NOZZLE USING CFD"** is being submitted by P.SAI TEJA (315126520177) , B.S.S.PRUDHVI (315126520193), P.RAMESH APPALARAJU (315126520179),T.V.R.S.S.N.PRUDHVI(315126520214) and P.SANDEEP (315126520167) in partial fulfilment of the requirements for the award of the degree with **BACHELOR OF TECHNOLOGY**, to the Department of Mechanical Engineering, Anil Neerukonda Institute Of Technology & Sciences (A) Visakhapatnam during the academic year 2015-2019.

Head of the Department
(Dr.B.NAGA RAJU)
Head of the Department
Dept. Of Mechanical Engineering
ANITS, Sangivalasa
Visakhapatnam.

S. Ramanjaneyulu
Project Guide
(S.RAMANJANEYULU)
Assistant Professor
Dept. Of Mechanical Engineering
ANITS, Sangivalasa
Visakhapatnam.

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCE
Sangivalasa-531 162, VISAKHAPATNAM Dist. A.P

THIS PROJECT WORK IS APPROVED BY THE FOLLOWING BOARD OF EXAMINERS

INTERNAL EXAMINER:

15.4.19

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES
Sangivalasa 531 162 VISAKHAPATNAM Dist A.P

EXTERNAL EXAMINER:

15/04/19

ACKNOWLEDGEMENT

We express immensely our deep sense of gratitude to **Mr.S.RAMANJANEYULU**, Assistant Professor, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences, Sangivalasa, Bheemunipatnam (mandal), Visakhapatnam District for his valuable guidance and encouragement at every stage of the work made it a successful fulfilment.

We were very thankful to our **Professor T.SUBRAHMANYAM**, Principal, ANITS and **Professor B. NAGA RAJU**, Head of the Department, Mechanical Department, Anil Neerukonda Institute of Technology & Sciences (A) for their valuable suggestions.

We express our sincere thanks to the members of non-teaching staff of Mechanical Engineering for their kind co-operation and support to carry on work.

Last but not least, we would like to convey our thanks to all who have contributed either directly or indirectly for the completion of work.

P SAI TEJA	**315126520177**
B.S.S.PRUDHVI	**315126520193**
P.RAMESH APPALARAJU	**315126520179**
T.R.V.S.S.N.PRUDHVI	**315126520214**
P.SANDEEP	**315126520167**

CONTENTS

Chapter 5: RESULTS AND DISCUSSIONS

ABSTRACT

Nozzle is a device which is used to give the direction to the gases coming out of the combustion chamber. Nozzle is a tube which has a capacity to convert the thermo-chemical energy generated in the combustion chamber into kinetic energy. The nozzle converts the low velocity, high pressure, high temperature gas in the combustion chamber into high velocity gas of lower pressure and low temperature. A convergent divergent nozzle is used if the nozzle pressure ratio is high. High performance engines in supersonic aircrafts generally incorporate some form of a convergent-divergent nozzle. Our analysis is carried using software like SolidWorks Module for designing of the nozzle and ANSYS Fluent 15.0 for analysing the flows in the nozzle. This thesis contains analysis over a convergent divergent rocket nozzle which is performed by varying the divergent angle of the convergent divergent nozzle and the contours of the pressure, velocity, temperature, density and Mach number are determined and optimized angle of the divergent angle of the convergent divergent nozzle is been determined for the rocket nozzle. The classical one-dimensional inviscid theory does not completely reveal the complex flow features in a convergent divergent nozzle accurately. The code fluent had been used to compute various flow parameters using an axisymmetric Convergent Divergent nozzle for different nozzle delivery pressures and angle of divergence.

LIST OF TABLES

LIST OF FIGURES

CHAPTER-I

INTRODUCTION

1.1 Importance of Nozzles in Aerospace Applications

Swedish engineer of French descent who, in trying to develop a more efficient steam engine, designed a turbine that was turned by jets of steam. The critical component – the one in which heat energy of the hot high-pressure steam from the boiler was converted into kinetic energy – was the nozzle from which the jet blew onto the wheel. De Laval found that the most efficient conversion occurred when the nozzle first narrowed, increasing the speed of the jet to the speed of sound, and then expanded again. Above the speed of sound (but not below it) this expansion caused a further increase in the speed of the jet and led to a very efficient conversion of heat energy to motion. The theory of air resistance was first proposed by Sir Isaac Newton in 1726. According to him, an aerodynamic force depends on the density and velocity of the fluid, and the shape and the size of the displacing object. Newton's theory was soon followed by other theoretical solution of fluid motion problems. All these were restricted to flow under idealized conditions, i.e. air was assumed to possess constant density and to move in response to pressure and inertia. Nowadays steam turbines are the preferred power source of electric power stations and large ships, although they usually have a different design-to make best use of the fast steam jet, de Laval's turbine had to run at an impractically high speed. But for rockets the de Laval nozzle was just what was needed

A nozzle (from nose, meaning 'small spout') is a tube of varying cross-sectional area (usually axisymmetric) aiming at increasing the speed of an outflow, and controlling its direction and shape. Nozzle flow always generates forces associated to the change in flow momentum, as we can feel by handholding a hose and opening the tap. In the simplest case of a rocket nozzle, relative motion is created by ejecting mass from a chamber backwards through the nozzle, with the reaction forces acting mainly on the opposite chamber wall, with a small contribution from nozzle walls. As important as the propeller is to shaft-engine propulsions, so it is the nozzle to jet propulsion, since it is in the nozzle that thermal energy (or any other kind of high-pressure energy source) transforms into kinetic energy of the exhaust, and its associated linear momentum producing thrust.

The flow in a nozzle is very rapid (and thus adiabatic to a first approximation), and with very little frictional loses (because the flow is nearly one-dimensional, with a favourable pressure gradient except if shock waves form, and nozzles are relatively short), so that the isentropic model all along the nozzle is good enough for preliminary design. The nozzle is said to begin where the chamber diameter begins to decrease (by the way, we assume the nozzle is axisymmetric, i.e. with circular cross-sections, in spite that rectangular cross-sections, said two-dimensional nozzles, are sometimes used, particularly for their ease of directionability). The meridian nozzle shape is irrelevant with the 1D isentropic model; the flow is only dependent on cross-section area ratios.

Real nozzle flow departs from ideal (isentropic) flow on two aspects:

- Non-adiabatic effects. There is a kind of heat addition by non-equilibrium radical-species recombination, and a heat removal by cooling the walls to keep the strength of materials in longduration rockets (e.g. operating temperature of cryogenic SR-25 rockets used in Space Shuttle is 3250 K, above steel vaporization temperature of 3100 K, not just melting, at 1700 K). Short duration rockets (e.g. solid rockets) are not actively cooled but rely on ablation; however, the nozzle-throat diameter cannot let widen too much, and reinforced materials (e.g. carbon, silica) are used in the throat region.
- There is viscous dissipation within the boundary layer, and erosion of the walls, what can be critical if the erosion widens the throat cross-section, greatly reducing exit-area ratio and consequently thrust.
- Axial exit speed is lower than calculated with the one-dimensional exit speed, when radial outflow is accounted for.

1.2 Types of nozzles

Types of nozzles are several types. They could be based on either speed or shape.

a. Based on speed the basic types of nozzles can be differentiated as
   - Spray nozzles
   - Ramjet nozzles

b. Based on shape the basic types of nozzles can be differentiated as
   - Conical
   - Bell
   - Annular

1.2.1 Spray Nozzle

Many nozzles produce a very fine spray of liquids.

- Atomizer nozzles are used for spray painting, perfumes, carburettors for internal combustion engines, spray on deodorants, antiperspirants and many other similar uses.
- Air-Aspirating Nozzle uses an opening in the cone shaped nozzle to inject air into a stream of water-based foam (CAFS/AFFF/FFFP) to make the concentrate "foam up". Most commonly found on foam extinguishers and foam handlines.

- Swirl nozzles inject the liquid in tangentially, and it spirals into the centre and then exits through the central hole. Due to the vortexing this causes the spray to come out in a cone shape.

1.2.2 Ram Jet Nozzle

Ramjet, sometimes referred to as a flying stovepipe or an athodyd (aero thermodynamic duct), is a form of airbreathing jet engine that uses the engine's forward motion to compress incoming air without an axial compressor or a centrifugal compressor. Because ramjets cannot produce thrust at zero airspeed, they cannot move an aircraft from a standstill. A ramjet-powered vehicle, therefore, requires an assisted take-off like a rocket assist to accelerate it to a speed where it begins to produce thrust. Ramjets work most efficiently at supersonic speeds around Mach 3 (2,300 mph; 3,700 km/h). This type of engine can operate up to speeds of Mach 6 (4,600 mph; 7,400 km/h).

Ramjets can be particularly useful in applications requiring a small and simple mechanism for high-speed use, such as missiles. Weapon designers are looking to use ramjet technology in artillery shells to give added range; a 120 mm mortar shell, if assisted by a ramjet, is thought to be able to attain a range of 35 km (22 mi). They have also been used successfully, though not efficiently, as tip jets on the end of helicopter rotors.

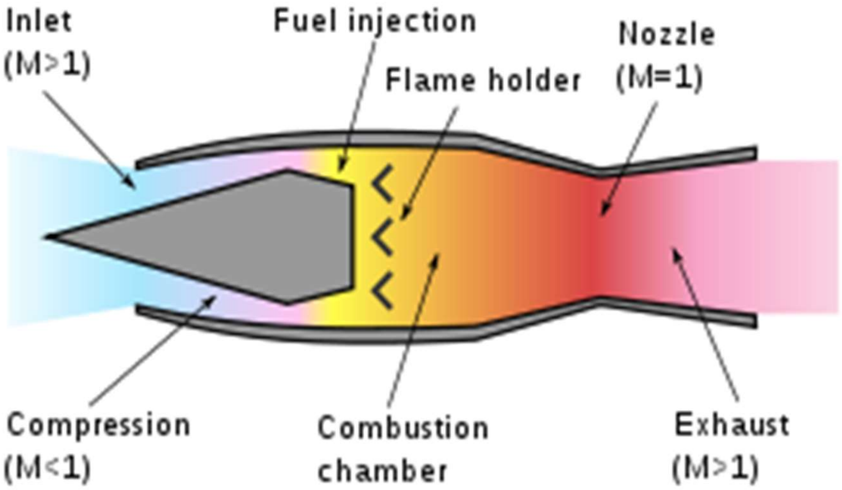


Fig 2.1 Ramjet Nozzle

1.2.3 Conical Nozzles

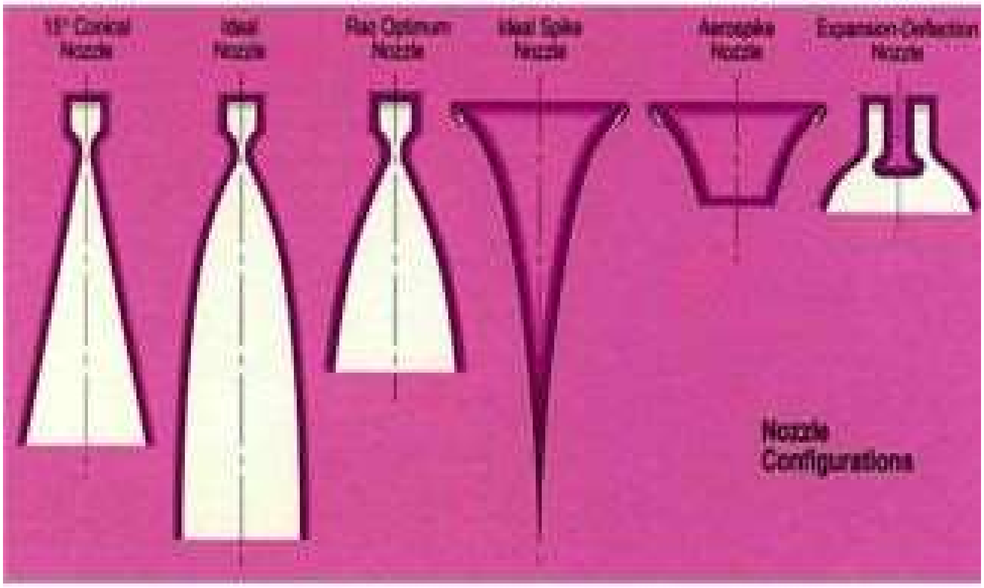


Fig 2.2 conical nozzles

1. Used in early rocket applications because of simplicity and ease of construction.
2. Cone gets its name from the fact that the walls diverge at a constant angle
3. A small angle produces greater thrust, because it maximizes the axial component of exit velocity and produces a high specific impulse
4. Penalty is longer and heavier nozzle that is more complex to build
5. At the other extreme, size and weight are minimized by a large nozzle wall angle – Large angles reduce performance at low altitude because high ambient pressure causes overexpansion and flow separation
6. Primary Metric of Characterization: Divergence Loss

1.2.4 Bell and Dual Bell

The Bell-shaped or contour nozzle is probably the most commonly used shaped rocket engine nozzle. It has a high angle expansion section (20 to 50 degrees) right behind the nozzle throat; this is followed by a gradual reversal of nozzle contour slope so that at the nozzle exit the divergence angle is small, usually less than a 10 degree half angle.

An ideal nozzle would direct all of the gases generated in the combustion chamber straight out the nozzle. That would mean the momentum of the gases would be axial, imparting the maximum thrust to the rocket. In fact, there are some non-axial components to the momentum. In terms of a momentum vector, there is an angle between the axis of the rocket engine and the gas flow. As a result, the thrust is lowered by varying amounts. The Bell or

Contour shape is designed to impart a large angle expansion for the gases right after the throat. The nozzle is then curved back in to give a nearly straight flow of gas out the nozzle opening. The contour used is rather complex. The large expansion section near the throat causes expansion shock waves. The reversal of the slope to bring the exit to near zero degrees causes compression shock waves. A properly designed nozzle will have these two sets of shock waves coincide and cancel each other out. In this way, the bell is a compromise between the two extremes of the conical nozzle since it minimizes weight while maximizing performance.

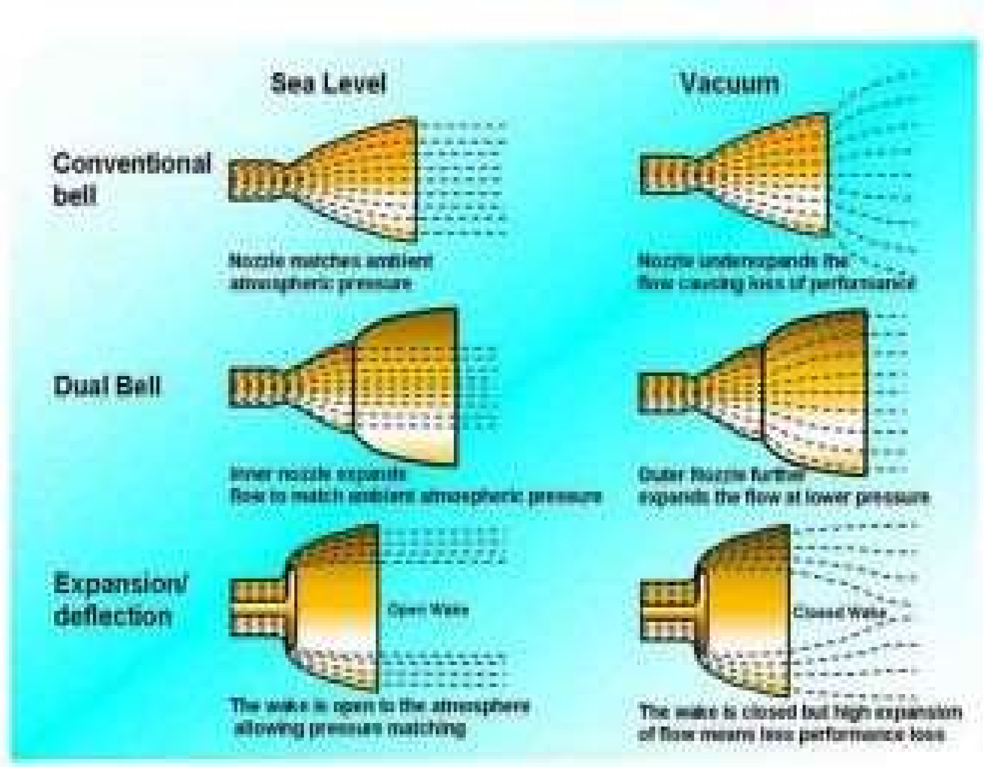


Fig 2.3: Bell and Dual Bell

1.3 Functions of Nozzle

The purpose of the exhaust nozzle is to increase the velocity of the exhaust gas before discharge from the nozzle and to collect and straighten the gas flow. For large values of thrust, the kinetic energy of the exhaust gas must be high, which implies a high exhaust velocity. The pressure ratio across the nozzle controls the expansion process and the maximum uninstalled thrust for a given engine is obtained when the exit pressure (P_e) equals the ambient pressure (P_0). The functions of the nozzle may be summarized by the following list:

1. Accelerate the flow to a high velocity with minimum total pressure loss.
2. Match exit and atmospheric pressure as closely as desired.
3. Permit afterburner operation without affecting main engine operation—requires

variable throat area nozzle.

4. Allow for cooling of walls if necessary.
5. Mix core and bypass streams of turbofan if necessary.
6. Allow for thrust reversing if desired.
7. Suppress jet noise, radar reflection, and infrared radiation (IR) if desired.
8. Two-dimensional and axisymmetric nozzles, thrust vector control if desired.
9. Do all of the above with minimal cost, weight, and boat tail drag while meeting life and reliability goals.

1.4 Conditions for operation

A de Laval nozzle will only choke at the throat if the pressure and mass flow through the nozzle is sufficient to reach sonic speeds, otherwise no supersonic flow is achieved, and it will act as a Venturi tube; this requires the entry pressure to the nozzle to be significantly above ambient at all times (equivalently, the stagnation pressure of the jet must be above ambient).

In addition, the pressure of the gas at the exit of the expansion portion of the exhaust of a nozzle must not be too low. Because pressure cannot travel upstream through the supersonic flow, the exit pressure can be significantly below the ambient pressure into which it exhausts, but if it is too far below ambient, then the flow will cease to be supersonic, or the flow will separate within the expansion portion of the nozzle, forming an unstable jet that may "flop" around within the nozzle, producing a lateral thrust and possibly damaging it.

In practice, ambient pressure must be no higher than roughly 2–3 times the pressure in the supersonic gas at the exit for supersonic flow to leave the nozzle.

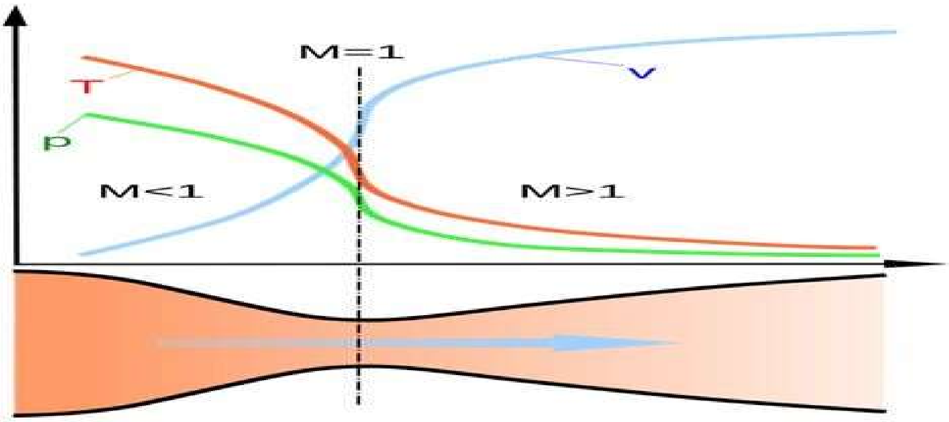


Fig 2.4 Mach number condition

Its operation relies on the different properties of gases flowing at subsonic and supersonic speeds. The speed of a subsonic flow of gas will increase if the pipe carrying it narrows because the mass flow rate is constant. The gas flow through a de Laval nozzle is isentropic (gas entropy is nearly constant). In a subsonic flow the gas is compressible, and sound will propagate through it. At the "throat", where the cross-sectional area is at its minimum, the gas velocity locally becomes sonic (Mach number = 1.0), a condition called choked flow. As the nozzle cross-sectional area increases, the gas begins to expand, and the gas flow increases to supersonic velocities, where a sound wave will not propagate backwards through the gas as viewed in the frame of reference of the nozzle (Mach number > 1.0).

1.5 Nozzle flow equations: -

Let us consider the steady isentropic 1D gas dynamics in a CD-nozzle, with the perfect gas model (i.e. pV= mRT and, taking T=0 K as energy reference, h =C_pT). Conservation of mass, momentum, and energy, in terms of the Mach number,

M = v/c (where c= $\sqrt{\gamma RT}$ stands for the sound speed),

become:

$$\dot{m} = \rho v A = \text{const} = \frac{p}{RT}\left(M\sqrt{\gamma RT}\right)A \quad \rightarrow \quad \frac{\mathrm{d}p}{p} - \frac{\mathrm{d}T}{2T} + \frac{\mathrm{d}M}{M} + \frac{\mathrm{d}A}{A} = 0$$

$$\rho v \mathrm{d}v = -\mathrm{d}p \rightarrow \frac{\mathrm{d}v^2}{2} + \frac{\mathrm{d}p}{\rho} = 0 \xrightarrow{dh = Tds + vdp} \frac{\mathrm{d}v^2}{2} + \mathrm{d}h - T\mathrm{d}s = 0 \xrightarrow{h_t = h + \frac{v^2}{2} = \text{const}} \mathrm{d}s = 0$$

$$h_t = h + \frac{v^2}{2} = \text{const} = c_p T + \frac{1}{2}M^2\gamma RT \quad \rightarrow \quad \frac{\mathrm{d}T}{T}\left(1 + \frac{\gamma - 1}{2}M^2\right) + (\gamma - 1)M\mathrm{d}M = 0$$

where logarithmic differentiation has been performed. Notice that, with this model, the isentropic condition can replace the momentum equation, so that differentiation of the isentropic relations for a perfect gas T/$P^{\frac{\gamma-1}{\gamma}}$=const, yields:

$$\frac{\mathrm{d}T}{T} = \frac{\gamma - 1}{\gamma}\frac{\mathrm{d}p}{p}$$

The energy balance (Δh_t=q+w) implies the conservation of total enthalpy and total temperature (h_t=$C_p T_t$), and the non-friction assumption implies the conservation of total pressure (pt), with the relations between total and static values given by:

$$\frac{T_t}{T} = 1 + \frac{v^2}{2c_pT} = 1 + \frac{\gamma - 1}{2}M^2 = \left(\frac{p_t}{p}\right)^{\frac{\gamma-1}{\gamma}}$$

Notice that, with the perfect gas model, γ remains constant throughout the expansion process. However, when the engine flow is composed of hot combustion products, real gas effects become important, and as the gas expands, γ shifts as a result of changes in temperature and in chemical composition. Maximum thrust is obtained if the gas composition is in chemical equilibrium throughout the entire nozzle expansion process. Choosing the cross-section area of the duct, A, as independent variable, the variation of the other variables can be explicitly found to be:

$$\left(1-M^2\right)\frac{dT}{T} = (\gamma - 1)M^2\frac{dA}{A}$$

$$\left(1-M^2\right)\frac{dp}{p} = \gamma M^2\frac{dA}{A}$$

$$\left(1-M^2\right)\frac{dM}{M} = -\left(1+\frac{\gamma-1}{2}M^2\right)\frac{dA}{A}$$

$$\left(1-M^2\right)\frac{dv}{v} = -\frac{dA}{A}$$

Equations above show that: - In converging section (dA< 0)

- When the flow is subsonic (M< 1→ (1-M^2 < 0): speed increases (dv>0), Mach number increases (dM>0), but pressure and temperature decreases.
- When the flow is supersonic (M>1 → (1-M^2 <0): speed decreases (d$v < 0$), Mach number decreases(dM<0), but pressure and temperature increases. In Diverging Section (dA>0):
- When the flow is subsonic (M< 1→ (1-M^2 < 0): speed decreases (d$v < 0$), Mach number decreases(dM<0), but pressure and temperature increases.
- When the flow is supersonic (M>1 → (1-M^2 <0): speed increases (dv>0), Mach number increases (dM>0), but pressure and temperature decreases.

1.6 Chocked Flow

Choked flow Choking is a compressible flow effect that obstructs the flow, setting a limit to fluid velocity because the flow becomes supersonic and perturbations cannot move upstream; in gas flow, choking takes place when a subsonic flow reaches M=1, whereas in liquid flow, choking takes place when an almost incompressible flow reaches the vapour pressure (of the main liquid or of a solute), and bubbles appear, with the flow suddenly jumping to M>1. Going on with gas flow and leaving liquid flow aside, we may notice that M=1 can only occur in a nozzle neck, either in a smooth throat where dA=0, or in a singular throat with discontinuous area slope (a kink in nozzle profile, or the end of a nozzle). Naming with a '*' variables the stage where M=1 (i.e. the sonic section, which may be a real throat within the nozzle or at some extrapolated imaginary throat downstream of a subsonic nozzle), and integrating from A to A*, equations become:

$$\frac{T^*}{T}=\frac{1+\frac{\gamma-1}{2}M^2}{\frac{\gamma+1}{2}} \xrightarrow{T_t=T\left(1+\frac{\gamma-1}{2}M^2\right)} \frac{T^*}{T_t}=\frac{2}{\gamma+1}$$

$$\frac{p^*}{p}=\left(\frac{1+\frac{\gamma-1}{2}M^2}{\frac{\gamma+1}{2}}\right)^{\frac{\gamma}{\gamma-1}} \xrightarrow{\frac{T}{T_t}=\left(\frac{p}{p_t}\right)^{\frac{\gamma-1}{\gamma}}} \frac{p^*}{p_t}=\left(\frac{2}{\gamma+1}\right)^{\frac{\gamma}{\gamma-1}}$$

$$\frac{A^*}{A}=M\left(\frac{\frac{\gamma+1}{2}}{1+\frac{\gamma-1}{2}M^2}\right)^{\frac{\gamma+1}{2(\gamma-1)}}$$

where the expressions for total temperature Tt and total pressure pt has been substituted to show that temperature and pressure at the throat (also known as critical values), are just a function of γ, since, for isentropic flows, total conditions do not change along the stream.

1.7 Applications of Convergent- Divergent Nozzle: -

Convergent-divergent(C-D) type of nozzles have a lot of application as a propelling nozzle in automobile and jets. Few examples of the application of convergent divergent type of nozzles in engineering are:

- In power plants especially used in the steam turbines.
- In Rocket propulsions for providing sufficient thrust to move upwards
- For supersonic gas turbine engine especially for air intake when the air requirement for the engine is high.
- Convergent Divergent nozzles are most widely used in water supplying pumps.

1.8 Objectives of the Thesis

The main objective of the thesis is to optimize the Rocket engine nozzle by varying the divergent angle of the nozzle for varying pressures.

1. The standard dimensions of the rocket nozzle are taken from the literature.
2. Design of the nozzle has been done using SolidWorks design module
3. Standard dimensions of the Rocket Nozzle are varied by keeping the length constant

and changing the diverging angles of the CONVERGENT DIVERGENT nozzle that is 11°, 13°, 14°, 15° and 17°.

4. Meshing and Domain creation for the inlet and outlet of the nozzle has been done in the Ansys Fluent Module.
5. The Analysis is carried at varying pressure from 5 bar – 10 bar and the results are obtained by taking the Pressure Contour, Velocity Contour, Temperature Contour and Mach Number contour.
6. The Results obtained are used for selecting the optimized angle of the divergence in the Convergent Divergent Nozzle.

CHAPTER-II

LITERATURE REVIEW

2.1 Literature Review

Nozzle is used to convert the chemical-thermal energy generated in the combustion chamber into kinetic energy. The nozzle converts the low velocity, high pressure, high temperature gas in the combustion chamber into high velocity gas of lower pressure and temperature. Swedish engineer of French descent who, in trying to develop a more efficient steam engine, designed a turbine that was turned by jets of steam. The critical component – the one in which heat energy of the hot high-pressure steam from the boiler was converted into kinetic energy – was the nozzle from which the jet blew onto the wheel. De Laval found that the most efficient conversion occurred when the nozzle first narrowed, increasing the speed of the jet to the speed of sound, and then expanded again. Above the speed of sound (but not below it) this expansion caused a further increase in the speed of the jet and led to a very efficient conversion of heat energy to motion. The theory of air resistance was first proposed by Sir Isaac Newton in 1726. According to him, an aerodynamic force depends on the density and velocity of the fluid, and the shape and the size of the displacing object. Newton" s theory was soon followed by other theoretical solution of fluid motion problems. All these were restricted to flow under idealized conditions, i.e. air was assumed to possess constant density and to move in response to pressure and inertia. Nowadays steam turbines are the preferred power source of electric power stations and large ships, although they usually have a different design-to make best use of the fast steam jet, de Lavals turbine had to run at an impractically high speed. But for rockets the de Laval nozzle was just what was needed. Many researchers have done the research on the nozzles for obtaining optimized conditions of which some of the research are as following: -

Bogdan-Alexandru Belega et-al [1] said that the Nozzle is a device designed to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that exhaust from them. Nozzles come in a variety of shapes and sizes depending on the mission of the rocket, this is very important for the understanding of the performance characteristics of rocket. By the proper geometrical design of the nozzle, the exhaust of the propellant gases will be regulated in such a way that maximum effective rocket velocity can be reached. Convergent divergent nozzle is the most commonly used nozzle since in using it the propellant can be heated in combustion chamber. After getting heated the propellant first converges at the throat of the nozzle and then expands under constant temperature in the divergent part. In the present paper, flow through the convergent divergent nozzle study is carried out by using a finite volume rewarding code, FLUENT 6.3. The nozzle geometry modeling and mesh generation has been done using GAMBIT 2.4 Software. Computational results are in good acceptance with the experimental results taken from the literature.

P. Vinod Kumar et-al [2] Nozzle is a device designed to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the Fluid that exhaust from them. Convergent-divergent nozzle is the most commonly used nozzle since in using it the propellant can be heated incombustion chamber. In this project we designed a new Tri-nozzle to increase the velocity of fluids flowing through it. It is designed based on basic convergent-Divergent nozzle to have same throat area, length, convergent angle and divergent angle as single nozzle. But the design of Tri-nozzle is optimized to have high expansion co-efficient than single nozzle without altering the divergent angle. In the present paper, flow through theTri-nozzle and convergent divergent nozzle study is carried out by using SOLID WORKS PREMIUM 2014.The nozzle geometry modeling and mesh generation has been done using SOLID WORKS CFD Software. Computational results are in goodacceptance with the experimental results taken from the literature.

Sudhir Singh Rajput et-al [3] said that the current research work is related to the computational fluid dynamic analysis of two-dimensional convergent-divergent nozzle in Ansys software. It using the CVM (control volume method) to solve the governing equation of fluid flow problem formulated under the given boundary condition. The basic aim of the current study is to determine the most suitable or optimum configuration of convergent-divergent angle in DC Nozzle. The parameter of a nozzle is taken according to the DC nozzle geometry. The different configuration has made by vary angle from 15 to 40 degree at the step of 5 degrees for Convergent angle and for divergent angle, it varies from 12.5 degrees to 20 degrees at the step of 2.5 degrees. The analysis was performed in the fluent workbench of ansys software. The input data for the nozzle is taken as the temperature of exhaust gas and pressure at the inlet. The output data is obtained by fluent in the form of temperature plot and pressure distribution and velocity gradient and Mach number are calculated for each combination.

Nikhil d et-al [4] said that de Laval nozzles are mechanical devices which are used to convert the thermal and pressure energy into useful kinetic energy. The values of temperature, pressure and velocity should be available at every section of the nozzle so as to design the nozzle shape, insulation and cooling arrangements. This paper aims at providing theoretical formulae to calculate the above. The validation of these formulae is carried out using the Computational Fluid Dynamics (CFD) software ANSYS Fluent.

Arjun Kundu et-al [5] A Converging – diverging gas nozzle is used to obtain supersonic speed .The project work is focused on analyzing the converging-diverginganannular nozzle with the help of computational fluid dynamics. A Nozzle of certain dimension is taken and it is modelled in Creo 2.0 parametric and then it is imported to ANSYS 14.0 Workbench and the analysis is carried out for two different exit diameters of the nozzle. The pressure, temperature and velocity were kept same in both the cases at inlet. The simulation has been done using ANSYS Workbench (CFX module) and the effect of exit diameter is studied. The result of simulation was different in each case. The increase in velocity, decrease in temperature and the drop-in pressure was more for nozzle with smaller exit diameter than nozzle for large exit diameter.

Raganar Larusson [6] Operating a convergent-divergent nozzle under overexpanded conditions can lead to supersonic flow separation in the divergent section of the nozzle. In this case, an attached oblique shock wave forms at the separation base. The sudden pressure rise across the shock wave can cause damaging lateral pressure forces, or side-loads, to act on the nozzle if the separation line is asymmetric. Such asymmetry can be caused by downstream instabilities stemming from turbulence, external excitation or periodic modes. The work reported in this thesis investigated the applicability of applying modal decomposition methods to supersonic nozzle flows. Axisymmetric RANS and URANS simulations of nozzle flows were investigated using the Arnoldi algorithm and Dynamic Mode Decomposition, respectively. The Arnoldi method relies on a linearized flow solver and has the advantage of being able to detect asymmetric modes on two dimensional grids. The DMD, however, is a snapshot-based algorithm that needs no explicit linearization of the flow dynamics. Results show that these methods can successfully be applied to supersonic nozzle flows with separation and strong shocks. For example, the Arnoldi method predicted a helical screeching mode with impressive accuracy and the DMD analysis of a perturbed two-dimensional URANS flow field was able to detect modes linked to transonic resonance. Finally, Detached Eddy Simulations (DES) on a separated flow inside a Truncated Ideal Contoured Nozzle were performed for two separate nozzle pressure ratios (NPRs). The simulated side-loads were lower than experimentally measured values but within the uncertainty range. A three-dimensional DMD analysis was made of the DES data and revealed a strong ovalization mode at the lower NPR and a helical mode which could be linked to a peak in side-load spectrum at the higher NPR.

Venkatesh et-al [7] Advances in rocket performance depend heavily upon improved and properly integrated propulsion system. This project provides a discussion about the design procedure of supersonic convergent-divergent (CONVERGENT DIVERGENT nozzle). The

CONVERGENT DIVERGENT nozzles both conical and contour are designed on an assumption of the isentropic flow of the perfect gas. The computer code which uses the method of characteristics and the stream function to define high efficiency nozzle profile for isentropic, inviscid, ir-rotational supersonic flows of any working fluid for any userdefined exit Mach number. The designed nozzle area ratio is compared to theoretical area ratios for the selected fluid and desired exit Mach number. The nozzle geometry obtained from the code is independently checked with the commercial Computational Fluid Dynamics (CFD) code. ANSYS-FLUENT has been used to simulate flow on nozzle to verify the isentropic flow.

R Ramesh Kumar et-al [8] Computational fluid dynamics is one of the parts of fluid mechanics and calculating numerical methods are solving the various flows of fluid. Two-dimensional models, which is to solve the flow rate of supersonic flow and analyzed the mathematical operations, which has solved the energy equations. This iteration process has analyzed and shown better results of temperature and static pressure, velocity, static temperature. The investigating results showing better temperature and velocity flows. The 2D nozzle line diagram is done in Ansys 14.5 geometry modular and calculated iteration process can be done in fluid flow. The nozzle is done and meshes using automatic method and sizing of different value of the meshing process. In the order to analyzed the Ansys fluent software and solved the flow process of the convergent-divergent iteration nozzle. Standard nozzle equation manually calculated and compared with analyzing the results.

Ehsan Eslamian et-al [9] Numerical study of multiphase swirl flow induced by helical insert inside the supersonic nozzle is investigated as applied in sandblasting systems. The finding of this research is crucial in improving the performance of abrasive blasting systems by reducing operation time and saving energy. Simulations are performed with Reynolds Stress Model turbulence modelling to capture anisotropy inside the flow. To simulate particles inside the nozzle, discrete phase model has been applied. The multiphase Eulerian method did not provide accurate results on modelling particles with small volume fraction (<10%), and also it was not capable of modelling particles interaction. The analyses are performed at three inlet pressures (2, 3 and 4 atm) with constant mass flow rate for particles. The results show that swirl effect increases the mixing feature of the flow, and therefore increases the cleaning area on a working surface. Furthermore, with swirl there was less reduction in maximum velocity compare with single phase simulations, hence particles will have greater speed at the exit of the nozzle.

Olivera P. Kostić et-al [10] Computational modeling of complex supersonic airflow patterns is one of the greatest challenges in the domain of CFD analyses. The paper presents initial steps in numerical analysis of such flow, generated by convergent divergent nozzle with Mach number M = 2.6 at nozzle exit. The aim was to achieve good agreements with available experimental data, obtained during supersonic wind tunnel tests at VTI Žarkovo institute, where nozzle thrust vectoring possibilities had been investigated using air as test fluid, by placing different types of obstacles at the exit section. Paper is focussed on free exit flow, and flow with one selected obstacle type. Using structured mesh for both cases, the RANS equations with k-ω SST turbulent model have been applied. After quantitative and qualitative comparisons with available experimental data, good agreements have been obtained, where CFD was also able to provide additional flow field data, not measured during experiments.

Omid Joneydi Shariatzadeh et-al [11] The converging-diverging nozzles play a significant role in a supersonic wind tunnel, where they draw air from a gas reservoir. Due to the back-pressure conditions through the convergent section, air reaches sonic conditions at throat. These conditions lead this stream to flow further through the divergent section where the flow Mach number increases. Manipulating the determinative variables such as area ratio and back pressure, the obtained Mach number may be regulated. In this work a comprehensive simulation of a flow in a typical supersonic converging-diverging nozzle has been reported. In the respective nozzle, flow suddenly contracts at a certain point and then expands after throat. All the simulation endeavours have been carried out by ANSYS FLUENT utilizing the mesh geometries previously and precisely accomplished in GAMBIT . The simulations have been conducted in either 2D or 3D domains to provide better comparative platform. Also, the influence of the turbulence model, differentiation and computational grid to the solution has been studied. Furthermore, the numerical comparison between CFD modelling results and corresponding available measured data has been presented. The comparison analysis of the data demonstrates an accurate enough coordination between the experimental data and the simulation results, which is applied more to the 3D endeavours than 2Ds.

Yesu Ratnam.Maddu et-al [12] Computational Fluid Dynamics (CFD) is a method which depends on various numerical analysis and algorithms for finding solutions and analysing the complex problems that concerns fluid flow and it is a tributary of Fluid Mechanics. The objective of the current project is to determine the variation of flow parameters like pressure, temperature, velocity and density is visualized using CFD. The simulation of shockwave through CFD is also executed. In the current work, Study of flow through the convergent divergent nozzle is carried out by using finite volume method. The modelling of the nozzle geometry is done in PTC Creo Parametric 3.0. The discretization process and the analysis are carried out in ANSYS FLUENT 16.0.

Ramji V et-al [13] This works centres on the design of a De Laval (convergent - Divergent) nozzle to accelerate the flow to supersonic or hypersonic speeds and computational analysis of the same. An initial design of the nozzle is made from the method of characteristics. The coding was done in MATLAB to obtain the contour of the divergent section for seven different exit Mach numbers viz. 3,3.5,4,4.5,5 and 5.5.To quantify variation in the minimum length of the nozzle divergent section with respect to the exit Mach number, a throat of constant height (0.005m)and width (0.05m) was chosen for all the design. The area exit required for each Mach no varying from 1 to 5.5 was plotted using isentropic relations and was also used to verify the exit area of the nozzle for each of those Mach numbers. An estimate of the exit pressure ratio is obtained by using isentropic and normal shock relations. With this exit pressure ratio, a more refined verification is done by computational analysis using ANSYS Fluent software for a contour nozzle with exit Mach number 5.5. The spalart Allmaras and k-epsilon model were used for turbulence modelling.

2.2 Summary

The above literatures reveal that there is a large quantum of literature available on the problem of achieving higher propulsions for the jet engines by varying the nozzle parameters. The optimized angle of the divergence in the CONVERGENT DIVERGENT nozzle is carried out by the analytical methods of various parameters. The literature provides us the basis for the carrying out the analysis on the nozzle flow by computational methods to know the accurate values of the parameters obtained and thus helps in selection of the optimized angle of divergence for the Convergent Divergent Nozzle.

CHAPTER – III

DESIGN AND MODELLING OF CONVERGENT DIVERGENT NOZZLE USING SOLID WORKS DESIGN MODULE

3.1 Background of SolidWorks

The Solid Works software is a solid modelling computer aided design (CAD) and computer aided engineering (CAE) software program that runs on Microsoft Windows. The SolidWorks is produced by the Dassault Systems— a subsidiary of Dassault Systems. Solid Works is currently used by over 2 million engineers and designers at more than 165,000. SOLIDWORKS Corporation was founded in December 1993 by Massachusetts Institute of Technology graduate Jon Hirschtick; Hirschtick used $1 million he had made while a member of the MIT Blackjack Team to set up the company. Initially based in Waltham, Massachusetts, USA, Hirschtick recruited a team of engineers with the goal of building 3D CAD software that was easy to use, affordable, and available on the Windows desktop. Operating later from Concord, Massachusetts, SOLIDWORKS released its first product SolidWorks 95, in 1995. In 1997 Dassault, best known for its CATIA CAD software, acquired SolidWorks for $310 million in stock.

SolidWorks is a solid modeller, and utilizes a parametric feature-based approach which was initially developed by PTC (Creo/Pro-Engineer) to create models and assemblies. The software is written on Parasolid-kernel. Parameters refer to constraints whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric parameters, such as line lengths or circle diameters, or geometric parameters, such as tangent, parallel, concentric, horizontal or vertical, etc. Numeric parameters can be associated with each other through the use of relations, which allows them to capture design intent.

Design intent is how the creator of the part wants it to respond to changes and updates. For example, you would want the hole at the top of a beverage can to stay at the top surface, regardless of the height or size of the can.

Features refer to the building blocks of the part. They are the shapes and operations that construct the part. Shape-based features typically begin with a 2D or 3D sketch of shapes such as bosses, holes, slots, etc. This shape is then extruded or cut to add or remove material from the part. Operation-based features are not sketch-based, and include features such as fillets, chamfers, shells, applying draft to the faces of a part, etc.

SolidWorks files (previous to version 2015) use the Microsoft Structured Storage file format. This means that there are various files embedded within each SLDDRW (drawing files), SLDPRT (part files), SLDASM (assembly files) file, including preview bitmaps and metadata sub-files. Various third-party tools (see COM Structured Storage) can be used to extract these sub-files, although the subfiles in many cases use proprietary binary file formats.

The SolidWorks software mainly comprises of three basic amenities that are:

- Part Design
- Assembly Design
- Drawing Design

I.*Part design*

First Section of 3D design is part design. Part design simply a 3D representation of a single design components.

II. Assembly design

An assembly design consists of two or more components assembled together at their respective work positions using parametric relations. In SolidWorks, these relations are called mates. These mates allow you to constrain the degrees of freedom of the components at their respective work positions.

All components are created as separate part documents, and then they are placed and referenced in the assembly as external components. In this type of approach, the components are created in the Part mode and saved as the (.sldprt) documents.

After creating and saving all components of the assembly, you need to start a new assembly document (.sldasm) and insert the components in it using the tools provided in the Assembly mode. After inserting the components, you can assemble

II-I. Assembly mates: After designing the parts, now they are assembled by using the mate technology. While doing assembly the mates are the main point to be remember. In Solid Works, mates can be applied using the Mate Property Manager. Choose the Mate button in the Assemble Command Manager or choose Insert > Mate from the. Select a planar face, curved face, axis, or a point on the first component and then select the entity from the second component; the selected entities will be highlighted. The names of the selected entities will be displayed in the Entities to Mate selection box of the Mate Selections rollout.

III. Drawing-

Drawing in Solid Works is done by the use of ISO standards. In the drawing section firstly the size of sheet is decided

3.2 Design and Modelling of the Nozzle

Step 1: The dimensions of the nozzle is taken from the literature K.P.S.Surya Narayana1 , K.Sadhashiva Reddy "Simulation of Convergent Divergent Rocket Nozzle using CFD Analysis" (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 13, Issue 4 Ver. I (Jul. - Aug. 2016)

Step 2: The 2-D outline is made in the SolidWorks module and mirror command is used for the completion of the nozzle along the axis

Step 3: The complete 3-D nozzle is made using the revolve feature along the axis.

Step 4: The various nozzles of different diverging angles are made by changing the divergent angle of the nozzle keeping the length.

3.2.1 Dimensional Parameters for designing Convergent Divergent Nozzle

The Geometry of the nozzle was created using SolidWorks module by using the following parameters shown in the table 3.1 and in the fig 3.1

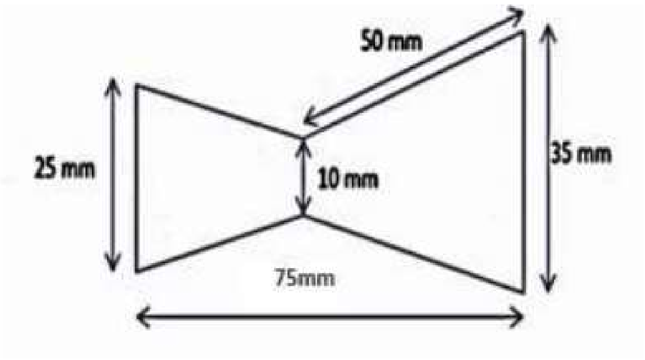


Fig 3.1 Outline drawing of convergent divergent nozzle

Table No 3.1 Design parameters

Parameter	Dimensional quantity
Inlet diameter	25mm
Throat diameter	10mm
Exit diameter	35mm
Divergent angle	11.6° ,12.5°,14.4°,15.6°,17.7°

3.2.2 Two-Dimensional Modelling of the Convergent Divergent nozzle

The designing of the Convergent Divergent nozzle is carried out with the help of the SolidWorks module method with standard dimensions from the literature. The standard model is being designed in the SolidWorks 2016 module by drawing the 2-D model of the convergent divergent nozzle with the known dimensions as shown:

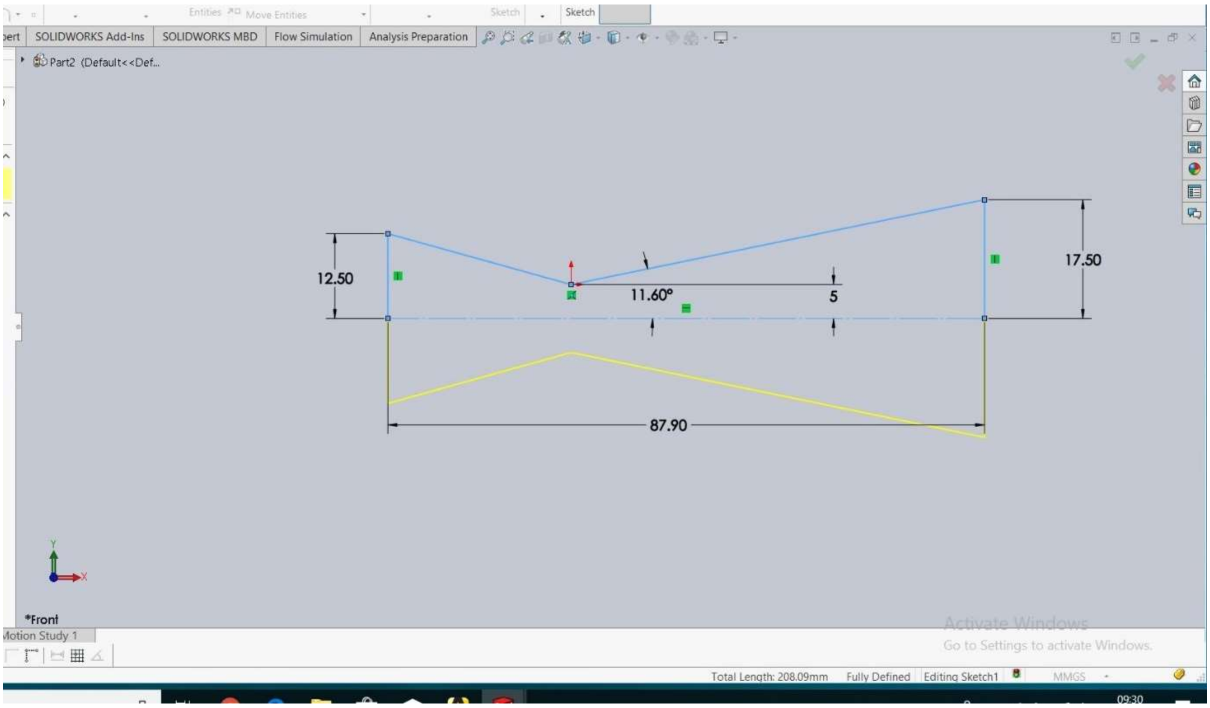


Fig 3.2 2-D design of the convergent divergent nozzle

3.2.3 Three-dimesional Modelling of the Convergent Divergent Nozzle

The dimensioning of the nozzle is given by the smart dimension command and the dimensions of the nozzle can be checked and now using the features command the design model is made to converted into 3-D model by revolve feature and made revolving around the centre line and the 3-D model of the de laval nozzle is made as shown in fig 3.3

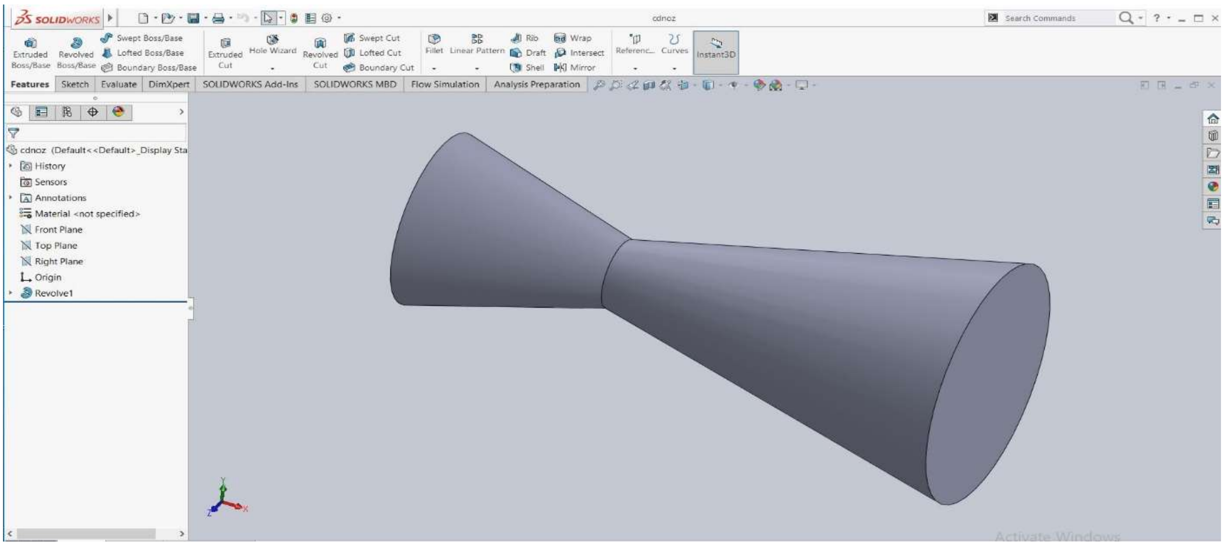
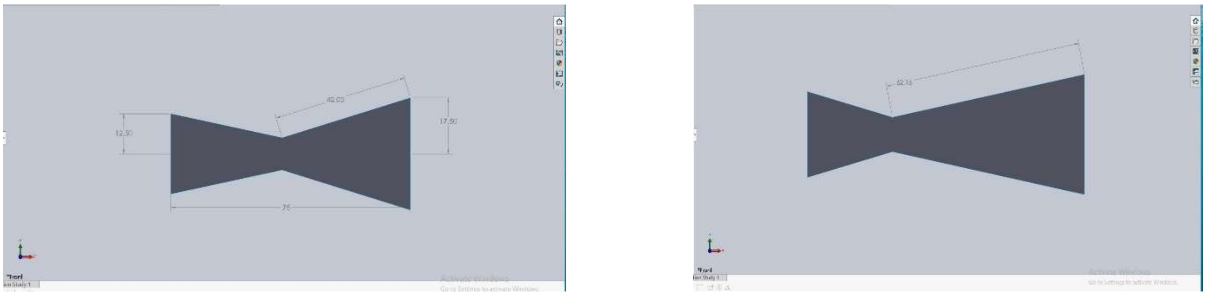


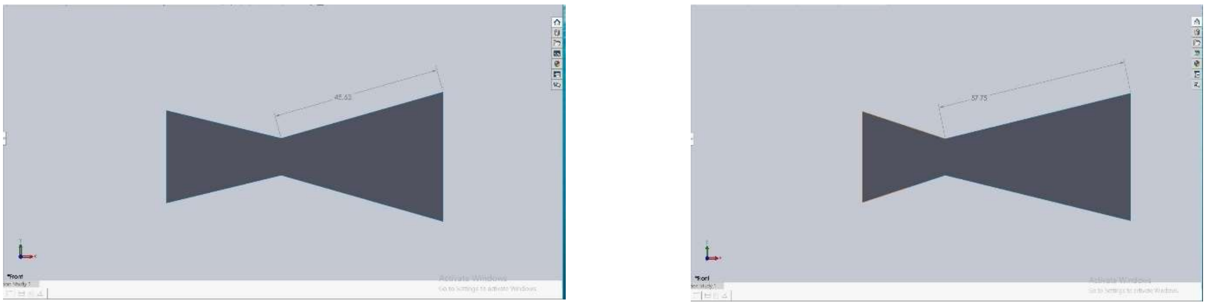
Fig 3.3 3-D model of the convergent divergent nozzle

3.2.4 Modelling of Convergent Divergent Nozzle with various divergent angle

The nozzle with the different divergent angle is made using the SolidWorks design software. The divergent angles of the nozzle are altered and four different nozzles are made by not altering the convergent angle and altering the divergent angle as shown in fig 3.4(a-b)



(a) Angle 12.5°   (b)Angle 14.4°



(c)   Angle 15.6°   (d)Angle 17.7°

Fig 3.4(a-b) Design of C-D nozzles having different divergent angle.

CHAPTER IV

CFD ANALYSIS OF THE CONVERGENT DIVERGENT NOZZLE

The Design of the Convergent Divergent Nozzle is made in the SolidWorks module : The 2-D outline is made in the SolidWorks module and mirror command is used for the completion of the nozzle along the axis. The complete 3-D nozzle is made using the revolve feature along the axis. The various nozzles of different diverging angles are made by changing the divergent angle of the nozzle keeping the length. The completed geometry is imported in the ANSYS FLUENT module and the analysis is carried out for different inputs of pressures and the parameters like Velocity, Temperature, Mach number and Density are studied from the results obtained.

4.1 FLUENT History

Analysis System known as ANSYS was founded in 1970 by Dr.John Swanson , who worked at the Westinghouse Nuclear Laboratories in Pittsburgh, he was responsible for the nuclear reactor and developed computer codes for obtaining the efforts in the rotor.

ANSYS did not start as CFD software but started as a CAE software and since 2003 begins with the acquisition of important software's such as CFX-4 later called CFX and ANSYS developed the new CFX and CFD software in the FLUENT module.

4.2 Scope of FLUENT

Simulation has become a linchpin of design in the current product development landscape. Engineers can use multi physics to better predict how designs will react to every conceivable environment in an effort to design faster, cheaper and better performing products. One platform which makes this possible is ANSYS Workbench which brings together their modelling, meshing, fluid, multi physics, structural, dynamics, electromagnetic and turbo system software components all under one roof. At the ANSYS Convergence Conference (ACC) they talked about the power of the Workbench and what might be in store for the future of ANSYS

4.3 Introduction to CFD

CFD is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyse problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interactions of fluids and gases with the complex surfaces used in engineering. However even with simplified equations and high speed supercomputers, only approximate solutions can be achieved in many cases. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic or turbulent flows are on ongoing area of research.

4.4 Working of CFD

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. Hence all codes contain three main elements.

1) Pre-Processing
2) Solver
3) Post-Processing

4.4.1 Pre-Processing

This is the first step in building and analysing a flow model. Pre-processor consists of input of a flow problem by means of an operator- friendly interface and subsequent transformation of this input into form of suitable for the use by the solver. The user activities at the Pre-Processing stage involve:

- Definition of the geometry of the region: The computational domain
- Grid generation the subdivision of the domain into a number of smaller, non-overlapping sub domains (or control volumes or elements selection of physical or chemical phenomena that need to be modelled.)
- Definition of Fluid properties.
- Specification of appropriate boundary conditions at cells which coincide with or touch the boundary. The solution of a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The accuracy of CFD solutions is governed by number of cells in the grid. In general, the larger numbers of cells better would be the solution accuracy. Both accuracy of the solution & its cost in terms of necessary computer hardware & calculations time are dependent on the fineness of the grid. Efforts are underway to develop CFD codes with adaptive meshing capability. Ultimately such programs will automatically refine the grid in areas of rapid variation.

4.4.2 Solver

The FLUENT CFD code has extensive interactivity, so we can make changes to the analysis at any time during the process. This saves time and enables to refine designs more efficiently Graphical User Interface (GUI) is intuitive, which helps to shorten the learning curve and make the modelling process faster. In addition, FLUENT’s adaptive and dynamic meshing capability is unique and works with a wide range of physical models. This capability makes it possible and simple to model complex moving objects in relation to flow. This solver provides the broadest range of rigorous physical models that have been validated against industrial scale applications, so we can accurately simulate real-world conditions, including multiphase flows, reacting flows, rotating equipment, moving and deforming objects, turbulence, radiation, acoustics and dynamic meshing. The FLUENT solver has repeatedly proven to be fast and reliable for a wide range of CFD applications. The speed to solution is faster because suite of software enables us to stay within one interface from geometry building through the solution process, to post-processing and final output.

4.4.3 Post Processing

This is the final step in CFD analysis and it involves the organization and interpretation of the predicted flow data and the production of CFD images and animations. FLUENT's software includes full post processing capabilities. FLUENT exports CFD's data to third-party post-processors and visualization tools such as Ensight , Fieldview and Techplot as well as to VRML formats . In addition, FLUENT CFD solutions are easily coupled with structural codes such as ABAQUS, MSC and ANSYS as well as to other engineering process simulation tools.

4.5 Analysis of the CONVERGENT DIVERGENT Nozzle

The designing of the Convergent Divergent nozzle is carried out with the help of the SolidWorks module method with standard dimensions from the literature. The standard model is being designed in the SolidWorks 2016 module by drawing the 3-D model of the convergent divergent nozzle with the known dimensions of various divergent angles and are been imported in the ANSYS FLUENT module and the analysis is carried out as the following procedure

Step 1: Import of the geometry of the 3-D nozzle in the geometry section of the FLUENT Module.

Step 2: Mesh generation of the input geometry is made in the module

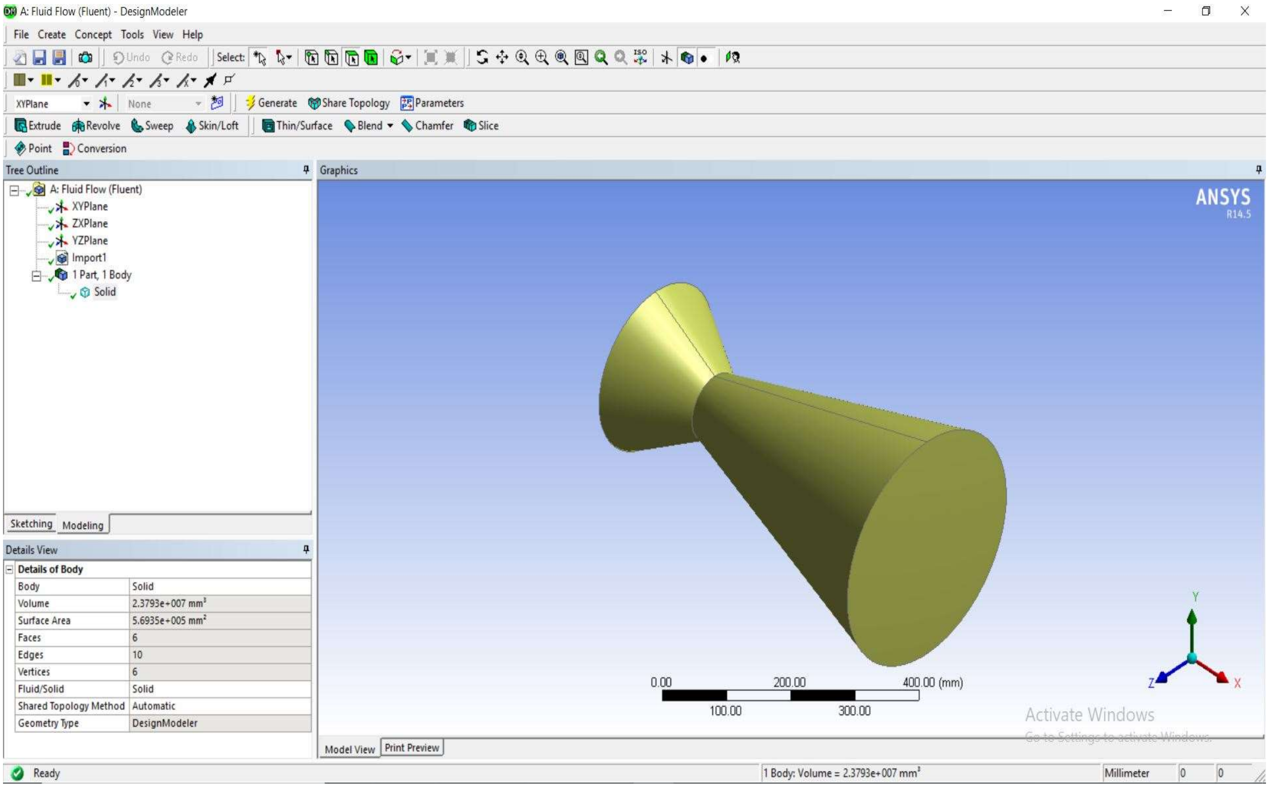
Step 3: General setup for the analysis such as type of feature, selection of model equation etc are carried out.

Step 4: Solution is made to run for the given number of iterations and the convergence of the solution is checked and the results are plotted.

Step 5: The results of the analysis carried out in the Setup feature are been plotted for various contours i.e., Pressure contour, Velocity Contour, Temperature Contour, Mach number Contour and Density Contour.

4.5.1 Import of the Geometry in ANSYS FLUENT Module

The Geometry created in the SolidWorks Module is imported in the ANSYS FLUENT Module by selecting the geometry in the FLUENT module and update the geometry as shown in the fig 4.1



Details View

Details of Body

Body	Solid
Volume	$2.3793e+007\ mm^3$
Surface Area	$5.6935e+005\ mm^2$
Faces	6
Edges	10
Vertices	6
Fluid/Solid	Solid
Shared Topology Method	Automatic
Geometry Type	DesignModeler

1 Body: Volume = $2.3793e+007\ mm^3$ Millimeter 0 0

Fig 4.1 Geometry Import in FLUENT Module

4.5.2 Mesh generation of the Geometry

- Create named selections as inlet, outlet and walls by using named selections feature and selecting the respective edges.
- Then insert edge sizing feature for each of the named selections and adjust the number of divisions on edge as 50.
- The mesh obtained initially will be unstructured mesh and cannot be used to obtain accurate results. Since the edges are prismatic the mesh can be converted into structured meshing by using Mapped Face Meshing. Now insert Mapped face meshing feature and select the target geometry for the mesh.
- Under Sizing : on proximity and curvature, Relevance : medium , Smoothening : high
- Finally, click on generate to create the mesh for the selected geometry.

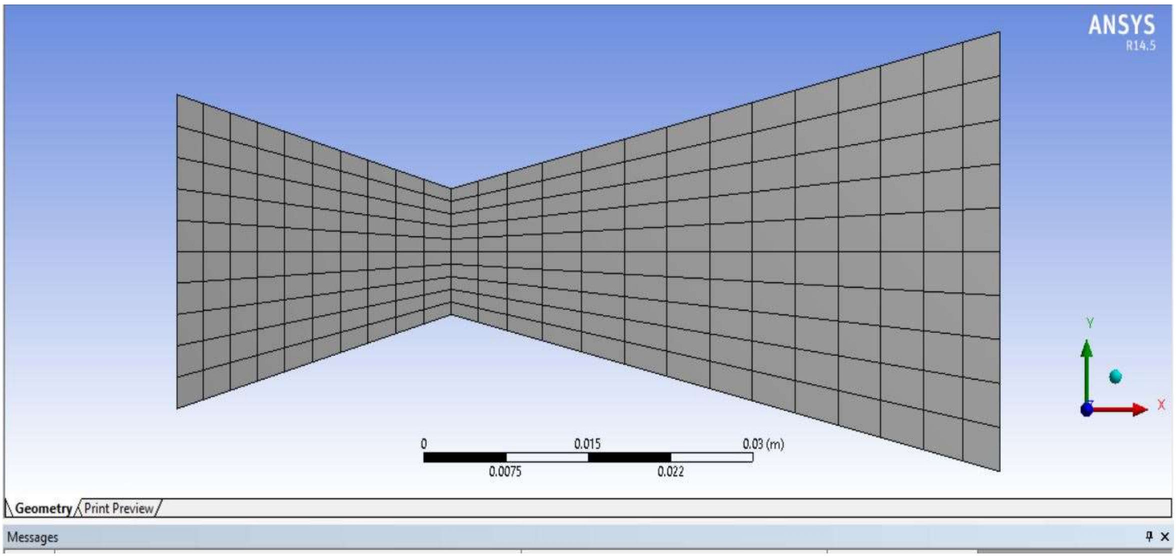


Fig 4.2 Mesh generation of the convergent divergent nozzle

4.5.3 Solution Setup in the ANSYS FLUENT Module

The Mesh generations of the Convergent Divergent Nozzle is done in the meshing part of the module and the solution conditions and setup conditions is given in the Solution option of the FLUENT Module in the following

Step 1: Select the General option and select the type as Density Based.

Step 2: Select the Model option and then select the Energy Equation to ON mode and change Viscous to Laminar.

Step 3: Select the Materials Option and select the Fluid type as Viscosity and Sutherland.

Step 4: Select the Cell Zone conditions and then the Fluid type is selected and the zone is selected on the Surface.

Step 5: Apply the Boundary Conditions, Select the Inlet and set the type Pressure Far Field and the inlet conditions as shown table 4.1 and in fig 4.3(a-c).

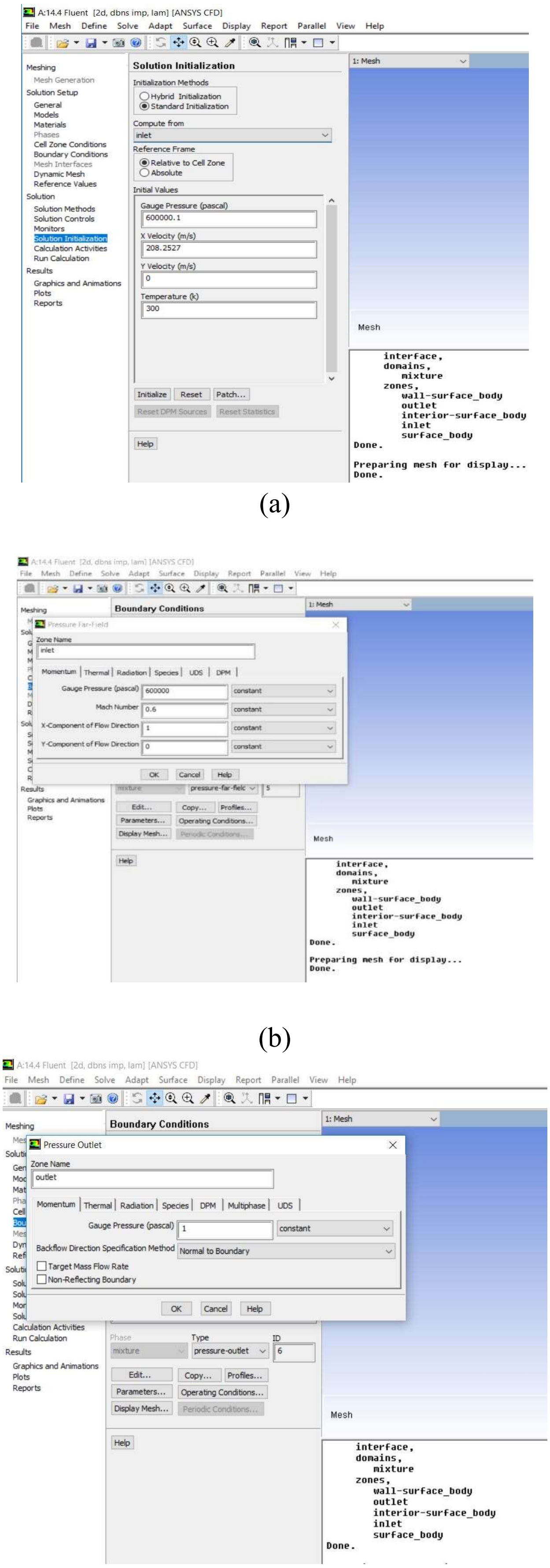
Table No 4.1 Inlet Boundary Conditions

Gauge Pressure	600000 Pascal
Mach Number	0.6
X-Component of Flow Direction	1
Y-Component Of Flow Direction	0

Step 6: Select the Outlet and give the conditions as shown in table no 4.2 and fig 4.3(a-c)

Table No 4.2 Outlet Boundary Conditions

Gauge Pressure	1
Backflow Direction Specification	Normal to Boundary



Solution Initialization

Initialization Methods
○ Hybrid Initialization
● Standard Initialization

Compute from: inlet

Reference Frame
● Relative to Cell Zone
○ Absolute

Initial Values
Gauge Pressure (pascal): 600000.1
X Velocity (m/s): 208.2527
Y Velocity (m/s): 0
Temperature (k): 300

```
      interface,
      domains,
         mixture
      zones,
         wall-surface_body
         outlet
         interior-surface_body
         inlet
         surface_body
Done.

Preparing mesh for display...
Done.
```

(a)

Boundary Conditions

Pressure Far-Field
Zone Name: inlet

Gauge Pressure (pascal): 600000 constant
Mach Number: 0.6 constant
X-Component of Flow Direction: 1 constant
Y-Component of Flow Direction: 0 constant

Phase: mixture Type: pressure-far-field ID: 5

```
      interface,
      domains,
         mixture
      zones,
         wall-surface_body
         outlet
         interior-surface_body
         inlet
         surface_body
Done.

Preparing mesh for display...
Done.
```

(b)

Boundary Conditions

Pressure Outlet
Zone Name: outlet

Gauge Pressure (pascal): 1 constant
Backflow Direction Specification Method: Normal to Boundary
☐ Target Mass Flow Rate
☐ Non-Reflecting Boundary

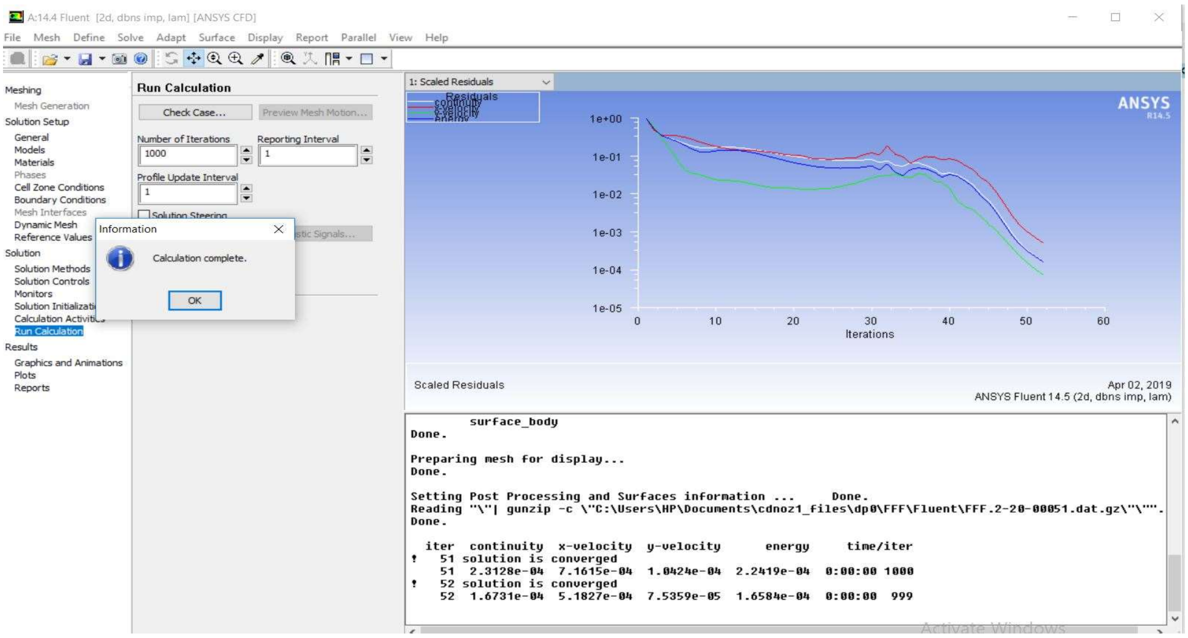
Phase: mixture Type: pressure-outlet ID: 6

```
      interface,
      domains,
         mixture
      zones,
         wall-surface_body
         outlet
         interior-surface_body
         inlet
         surface_body
Done.
```

(c)

Fig 4.3 Boundary conditions setup procedure

Step 7 : The boundary conditions are given as mentioned in the Step 5 and Step 6 and now initialize the solution as standard and calculate from the Inlet zone and set the number of Iterations as 1000 and run the solution.



Run Calculation

Number of Iterations: 1000

Reporting Interval: 1

Profile Update Interval: 1

Information

Calculation complete.

```
        surface_body
Done.

Preparing mesh for display...
Done.

Setting Post Processing and Surfaces information ...     Done.
Reading "\"| gunzip -c \"C:\Users\HP\Documents\cdnoz1_files\dp0\FFF\Fluent\FFF.2-20-00051.dat.gz\"\"".
Done.

  iter  continuity  x-velocity  y-velocity      energy     time/iter
!   51 solution is converged
    51  2.3128e-04  7.1615e-04  1.0424e-04  2.2419e-04  0:00:00 1000
!   52 solution is converged
    52  1.6731e-04  5.1827e-04  7.5359e-05  1.6584e-04  0:00:00  999
```

Fig 4.4 Post Processing Setup Calculations

4.5.4 Solution Calculations and Results

The Solution is made to run for the given number of iterations and the convergence of the solution is checked and the results are plotted. The results of the analysis carried out in the Setup feature are been plotted for various contours i.e., Pressure contour, Velocity Contour, Temperature Contour, Mach number Contour and Density Contour.

CHAPTER-V

RESULTS AND DISCUSSIONS

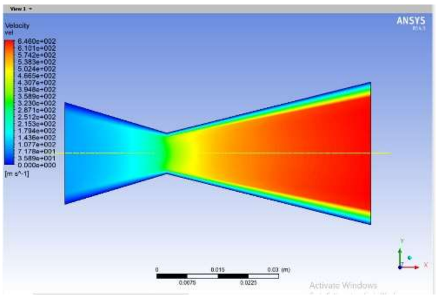
The Design and Modelling of the Convergent Divergent Nozzle is done in the SolidWorks Module and the CFD simulation of flow through Convergent Divergent nozzle is carried out with the input parameters and the pressure contour, velocity contour and Mach number contour is obtained for various angles of the divergent angle of the nozzle and the effect of the divergence angle in the Convergent Divergent Nozzle is observed.

5.1 Effect of divergence angle on the convergent divergent nozzle

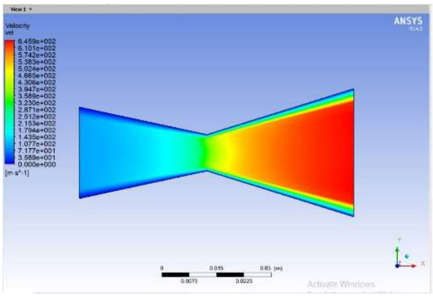
The Divergence angle of the Convergent Divergent Nozzle is varied from the standard dimensions obtained from the literature and the flow analysis is carried out in the ANSYS FLUENT Module for varying input pressures i.e., at 3 bar, 6 bar,9bar,12 bar and 15 bar. The contours of the Velocity, Temperature, Mach number and Density are obtained from the analysis and the effect of these change in Divergence Angle is studied by plotting the graphs between the output parameters i.e., Velocity, Temperature, Mach No, Density and Angle of Divergence.

5.1.1 Effect of divergence angle on the convergent divergent nozzle at 3 bar pressure inlet

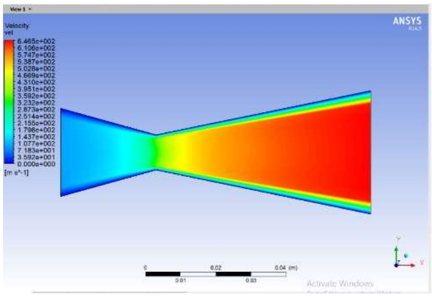
The Velocity Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.1(a-b) at the input condition Pressure of 3 bar.



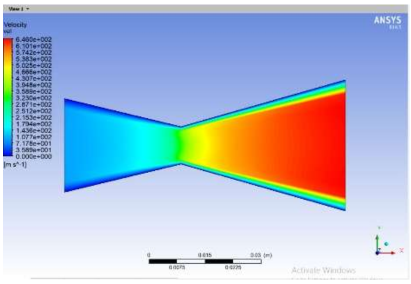
(a) Angle 11.6°



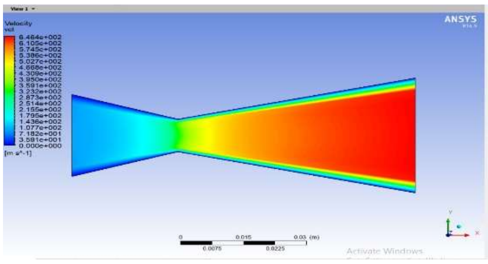
(b) Angle 12.5°



(c) Angle 14.4°



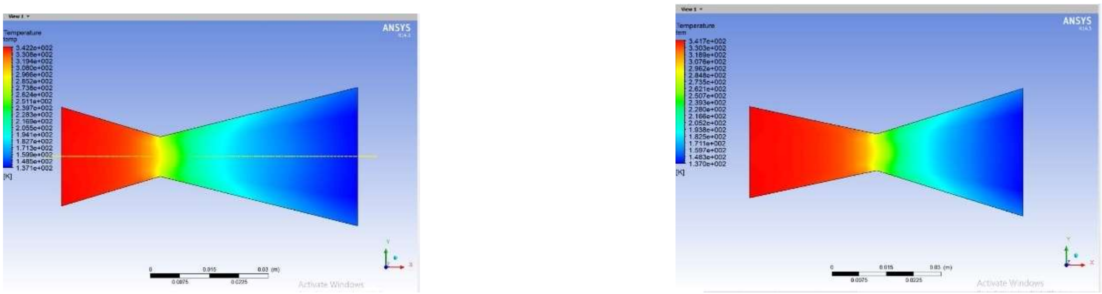
(d)Angle 15.6°



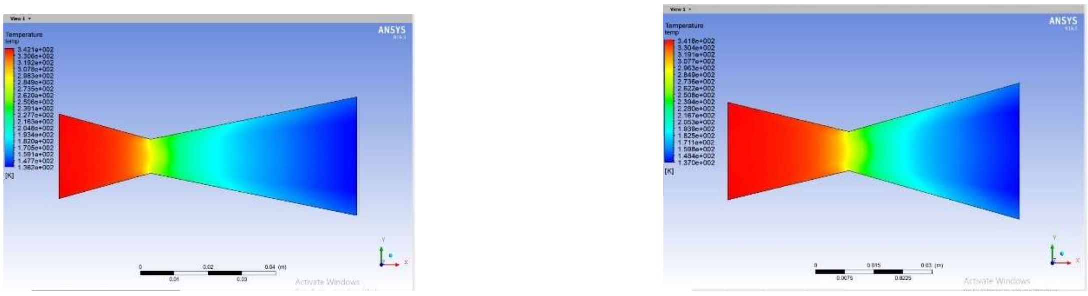
(e)Angle17.7º

Fig No 5.1 Velocity contours for various divergence angles at 3 bar pressure

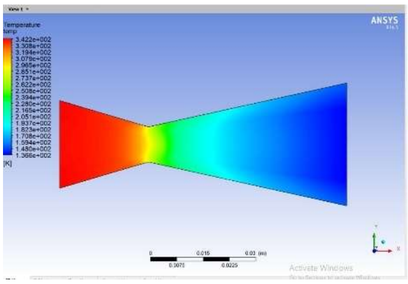
The Temperature Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.1(a-b) at the input condition Pressure of 3 bar.



(a) Angle 11.6º **(b)Angle 12.5º**



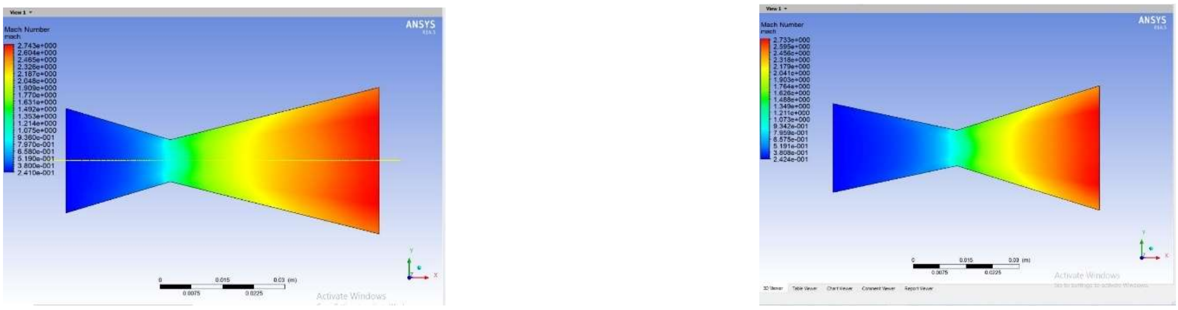
(c)Angle 14.4º **(d)Angle 15.6º**



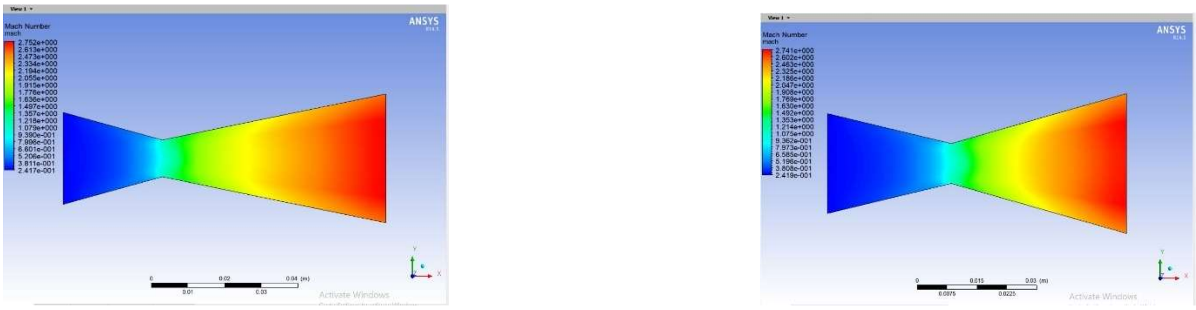
(e)Angle 17.7º

Fig No 5.2 Temperature contours for various divergence angles at 3 bar pressure

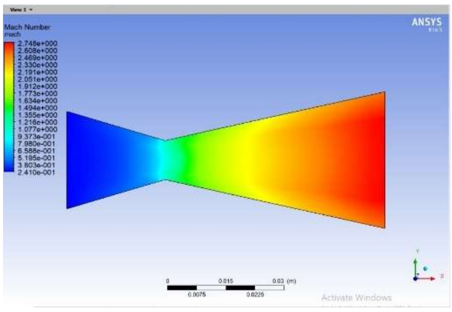
The Mach number Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.1(a-b) at the input condition Pressure of 3 bar.



(a) Angle 11.6^o (b)Angle 12.5^o



(c)Angle 14.4^o (d)Angle 15.6^o

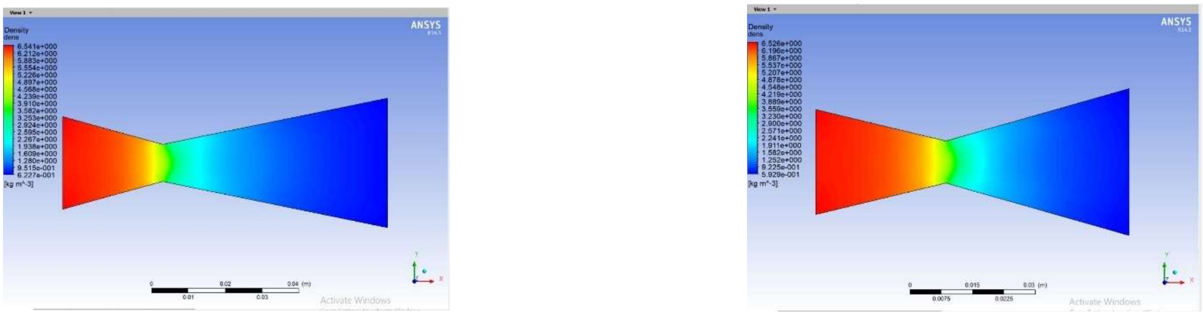


(e)Angle 17.7^o

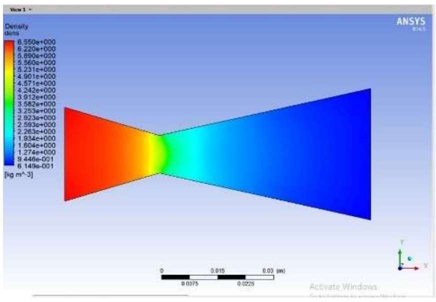
Fig No 5.3 Mach Number contours for various divergence angles at 3 bar pressure

The Density Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.1(a-b) at the input condition Pressure of 3 bar.

(a) Angle 11.6^o (b)Angle 12.5^o



(c)Angle 14.4^o (d)Angle 15.6^o

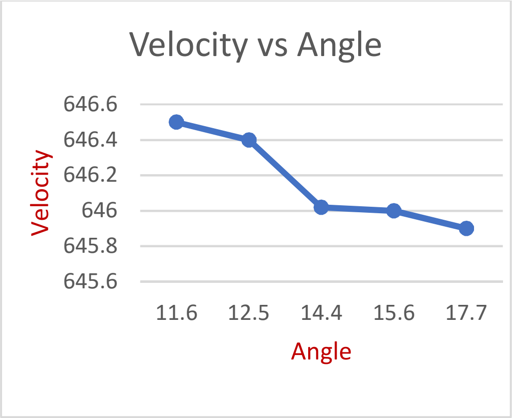


(e)Angle 17.7^o

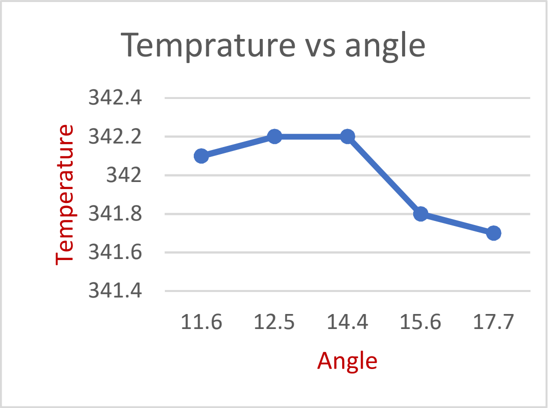
Fig No 5.4 Density contours for various divergence angles at 3 bar pressure

Table No 5.1 Comparison of divergent angles with various parameters of convergent divergent nozzle at 3 bar pressure

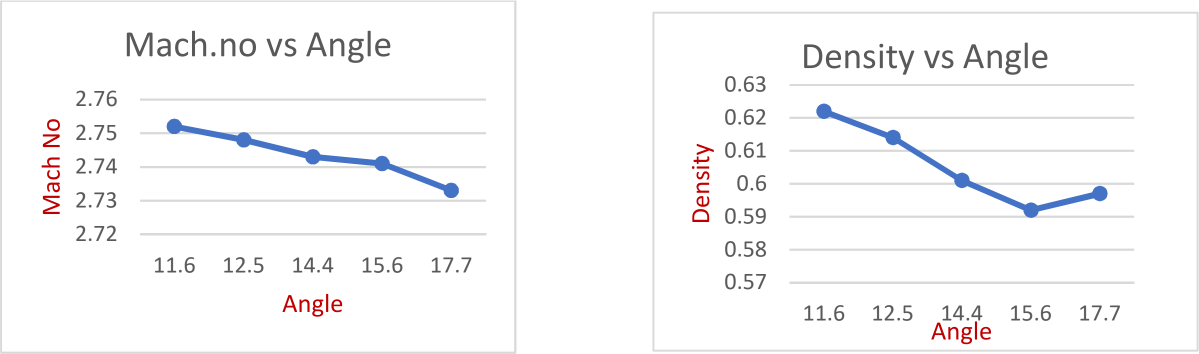
Angle	Inlet Pressure	Velocity	Temperature	Mach No	Density
11.6	3	646.5	342.1	2.752	0.622
12.5	3	646.4	342.2	2.748	0.614
14.4	3	646.02	342.2	2.743	0.601
15.6	3	646	341.8	2.741	0.592
17.7	3	645.9	341.7	2.733	0.597



(a)



(b)



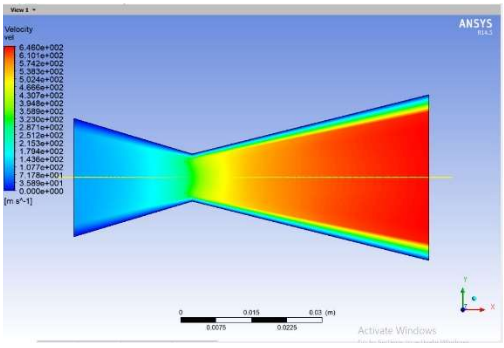
(c) **(d)**

Fig No 5.5 Graphs for various parameters vs divergence angles at 3 bar pressures

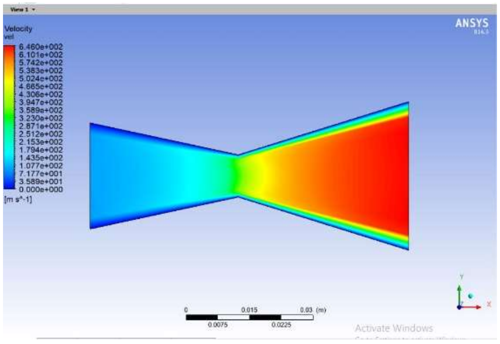
- The maximum velocity of flow through convergent divergent nozzle is obtained with the C-D Nozzle having a divergence angle of 11.6° and there is equal velocity obtained for the nozzles having divergence angle of 14.4° and 15.6° and the minimum velocity of flow is obtained for the nozzle having divergence angle of 17.7°.
- The Temperature is minimum at the nozzle having divergence angle of 17.7° and equal temperatures are attained for both the nozzles having divergence angle of 12.5° and 14.4° and the maximum temperature is obtained for the nozzle having divergence angle of 14.4°.
- The Mach Number is maximum for the nozzle having divergence angle of 11.6° and minimum ach number is for the nozzle having divergence angle of 17.7°.
- The Density is minimum for the C-D nozzle having divergence angle 15.6° and maximum for the nozzle having the divergence angle of 11.6°.

5.1.2 Effect of divergence angle on the convergent divergent nozzle at 6 bar pressure inlet

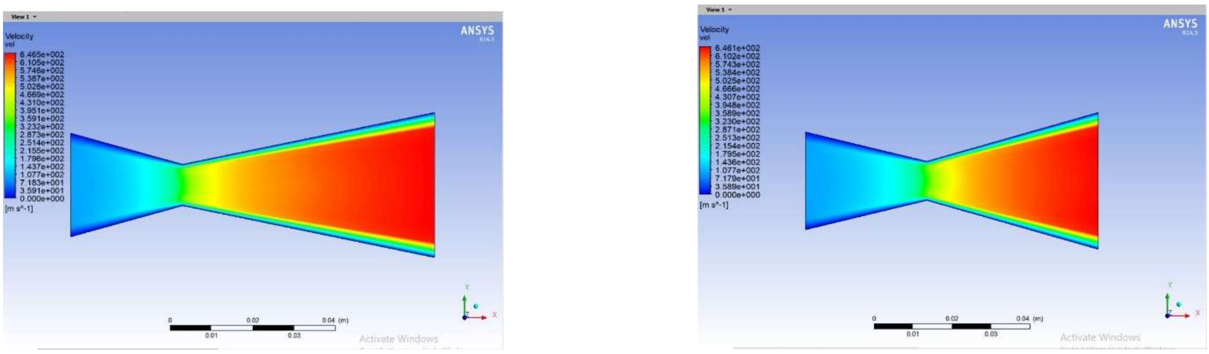
The Velocity Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 6 bar.



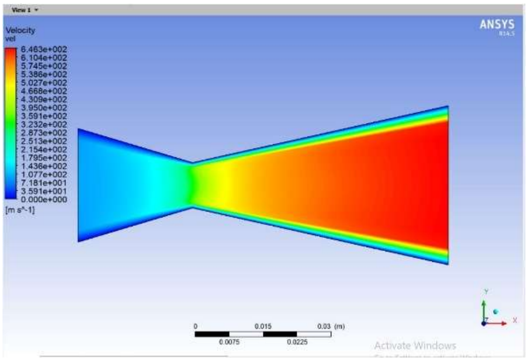
(a) Angle 11.6°



(b)Angle 12.5°



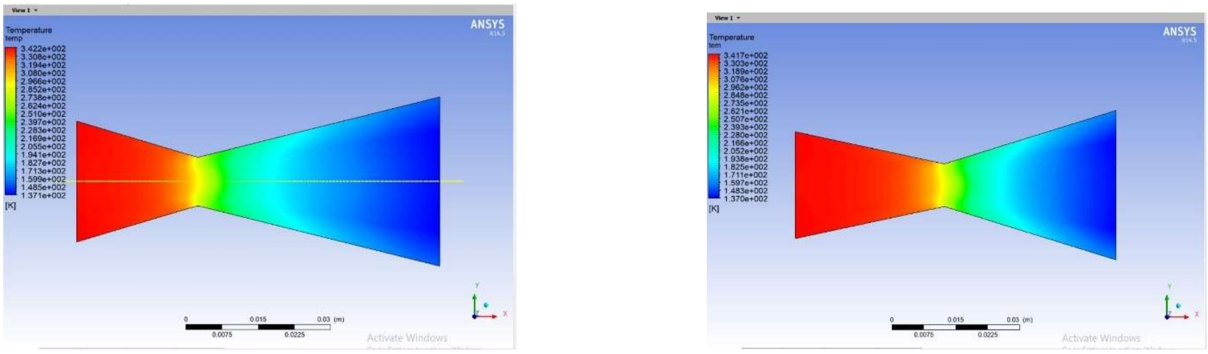
(c)Angle 14.4° **(d)Angle 15.6°**



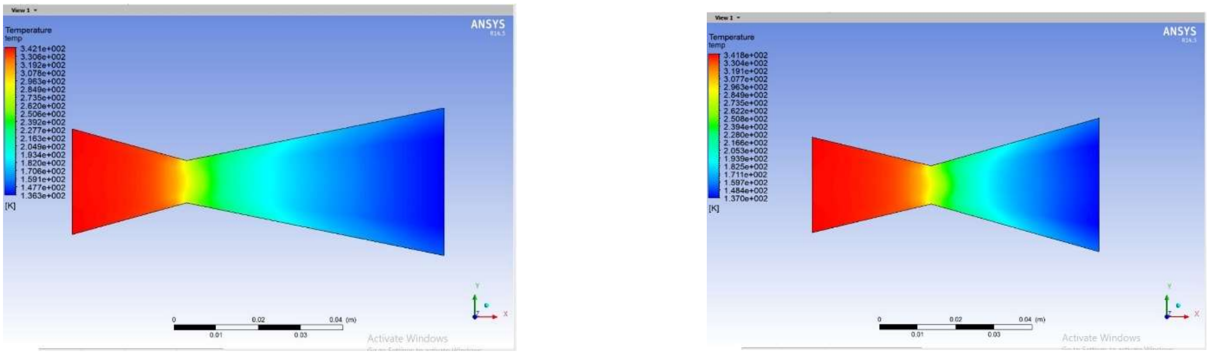
(e)Angle 17.7°

Fig No 5.6 Velocity contours for various divergence angles at 6 bar pressure

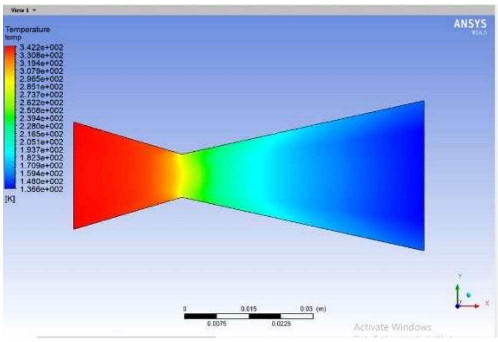
The Temperature Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 6 bar.



(a) Angle 11.6° **(b)Angle 12.5°**



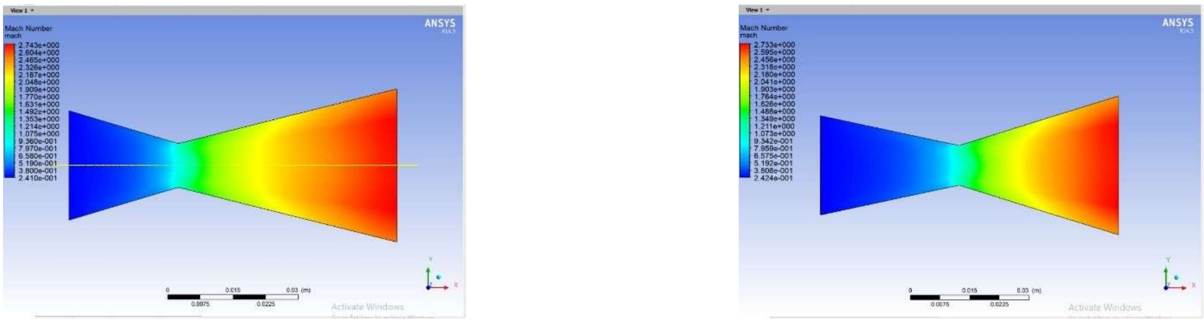
(c)Angle 14.4° **(d)Angle 15.6°**



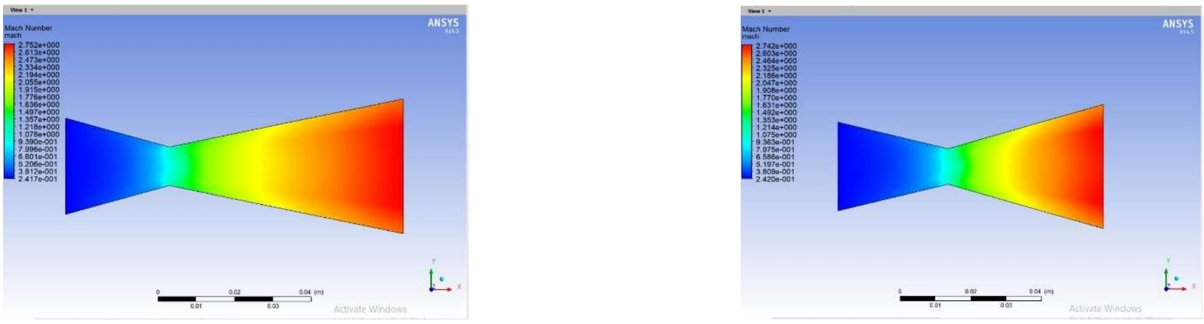
(e)Angle 17.7°

Fig No 5.7 Temperature contours for various divergence angles at 6 bar pressure

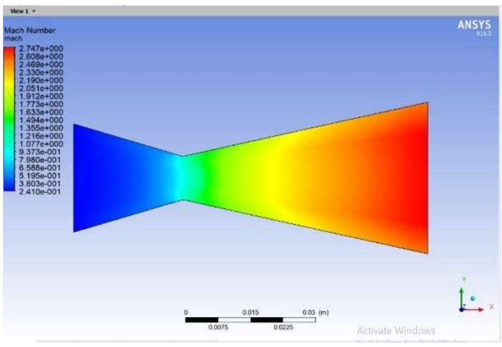
The Mach number Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 6 bar.



(a) Angle 11.6° **(b)Angle 12.5°**



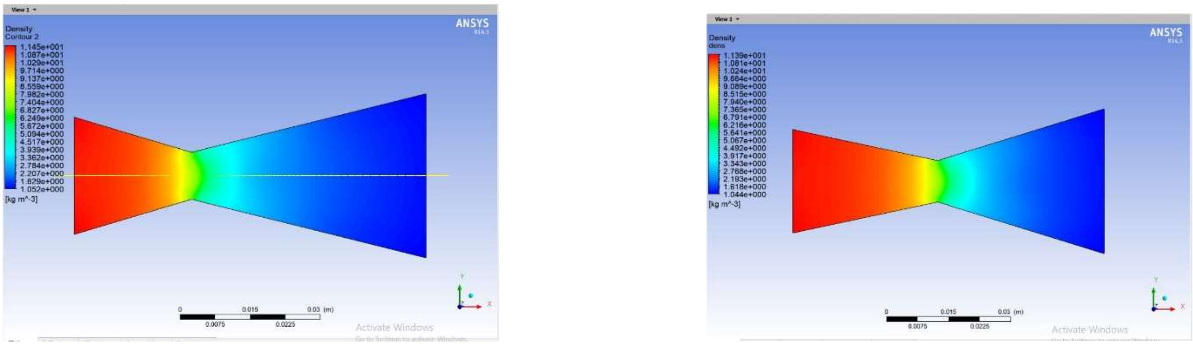
(c)Angle 14.4° **(d)Angle 15.6°**



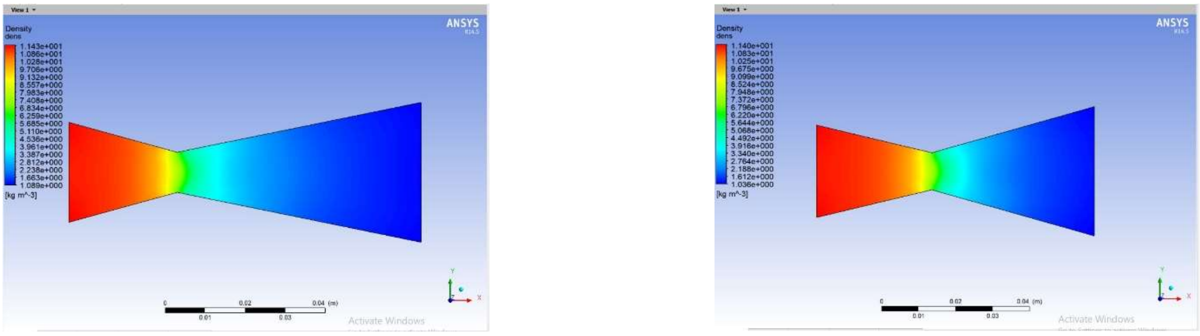
(e)Angle 17.7°

Fig No 5.8 Mach Number contours for various divergence angles at 6 bar pressure

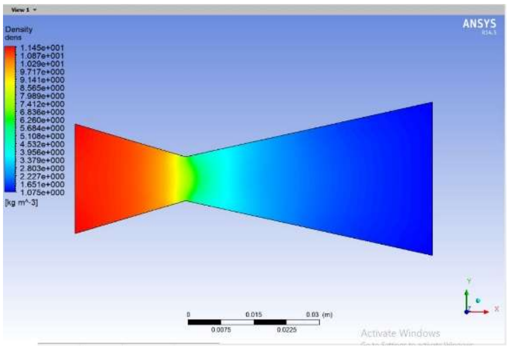
The Density Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 6 bar.



(a) Angle 11.6° **(b)Angle 12.5°**



(c)Angle 14.4° **(d)Angle 15.6°**

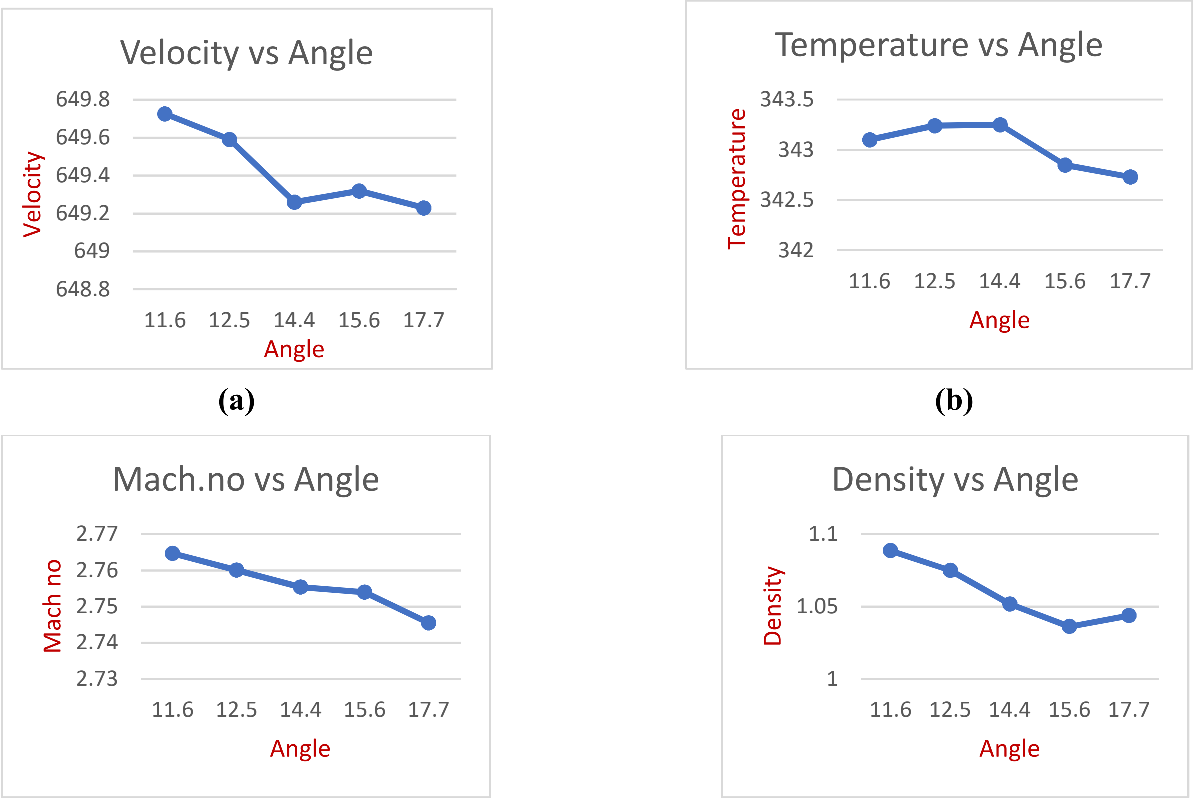


(e)Angle 17.7°

Fig No 5.9 Density contours for various divergence angles at 6 bar pressure

Table No 5.2 comparison of divergent angles with various parameters of convergent divergent nozzle at 6 bar pressure

Angle	Inlet Pressure	Velocity	Temperature	Mach No	Density
11.6	6	649.726	343.1	2.7647	1.08859
12.5	6	649.59	343.24	2.7601	1.07494
14.4	6	649.26	343.25	2.7554	1.05174
15.6	6	649.32	342.85	2.754	1.03617
17.7	6	649.23	342.73	2.7455	1.0438



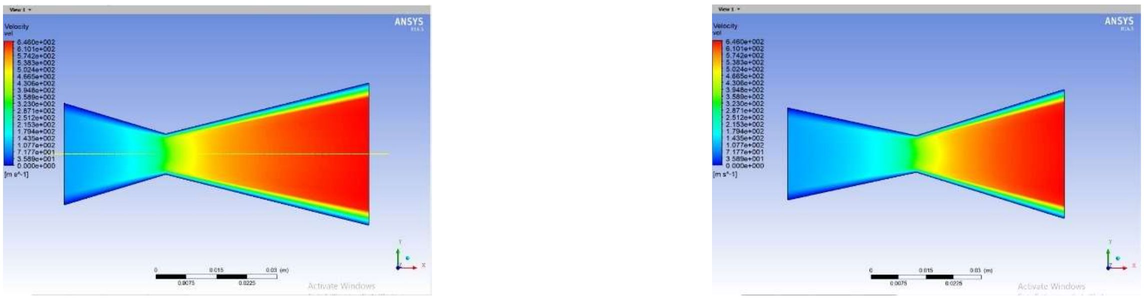
(a) **(b)** **(c)** **(d)**

Fig 5.10 Graphs for various parameters vs divergence angles at 6 bar pressures

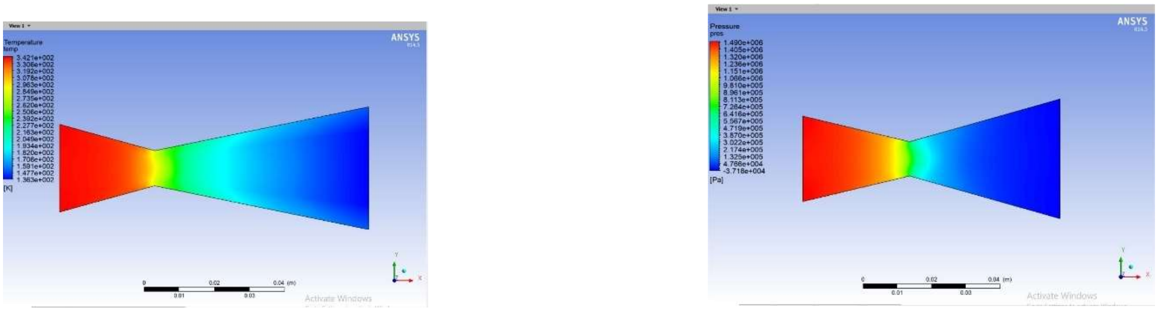
- The maximum velocity of flow through convergent divergent nozzle is obtained with the C-D Nozzle having a divergence angle of 11.6° and there is an increase in velocity obtained from the nozzles having divergence angle of 14.4° and 15.6° and the minimum velocity of flow is obtained for the nozzle having divergence angle of 17.7°.
- The Temperature is minimum at the nozzle having divergence angle of 17.7° and equal temperatures are attained for both the nozzles having divergence angle of 12.5° and 14.4° and the maximum temperature is obtained for the nozzle having divergence angle of 14.4°.
- The Mach Number is maximum for the nozzle having divergence angle of 11.6° and minimum Mach number is for the nozzle having divergence angle of 17.7°.
- The Density is minimum for the C-D nozzle having divergence angle 15.6° and maximum for the nozzle having the divergence angle of 11.6°.

5.1.3 Effect of divergence angle on the convergent divergent nozzle at 9 bar pressure inlet

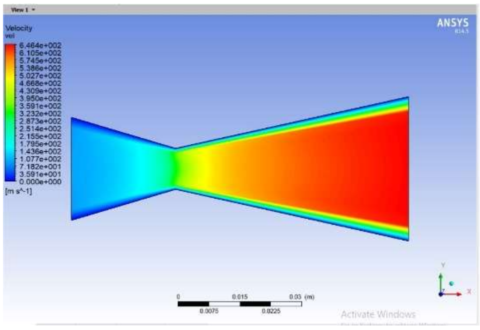
The Velocity Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 9 bar.



(a) Angle 11.6° **(b)Angle 12.5°**



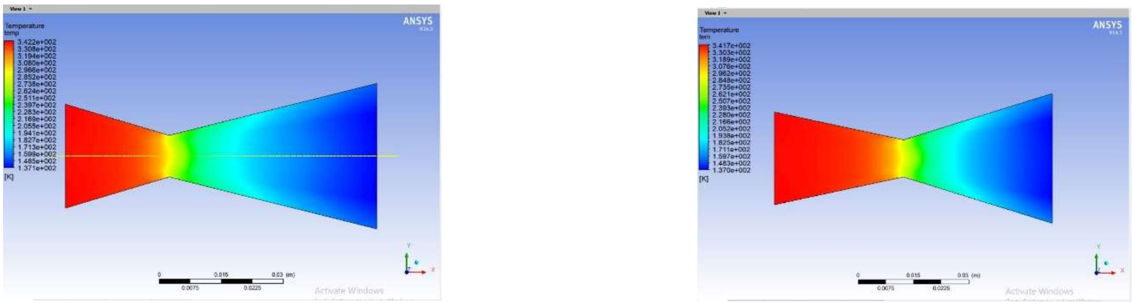
(c)Angle 14.4° **(d)Angle 15.6°**



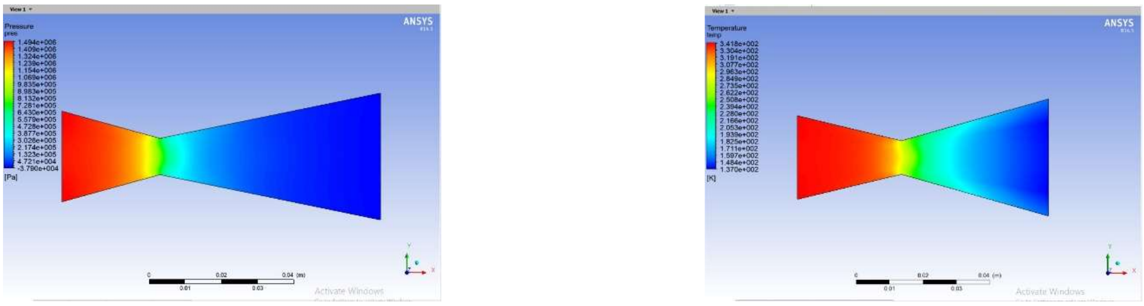
(e)Angle 17.7°

Fig No 5.11 Velocity contours for various divergence angles at 9 bar pressure

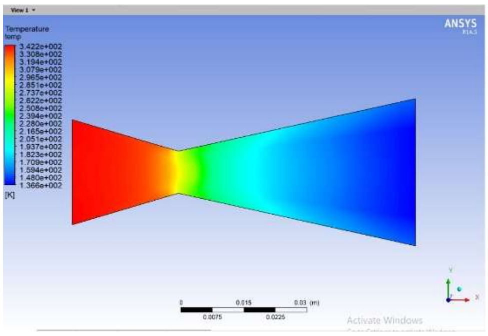
The Temperature Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 9 bar.



(a) Angle 11.6° **(b)Angle 12.5°**



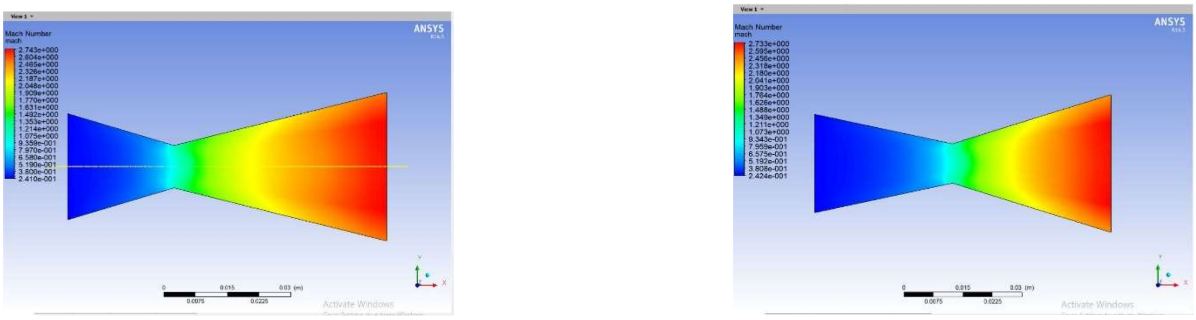
(c)Angle 14.4° **(d)Angle 15.6°**



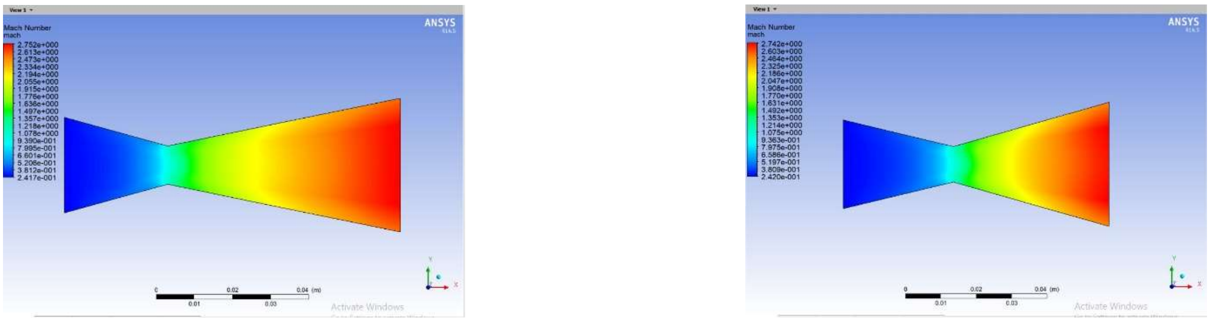
(e)Angle 17.7°

Fig No 5.12 Temperature contours for various divergence angles at 9 bar pressure

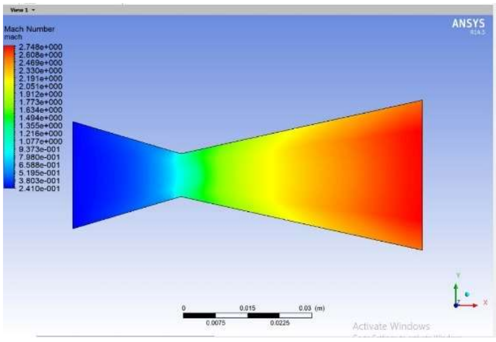
The Mach Number Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 9 bar.



(a) Angle 11.6° **(b)Angle 12.5°**



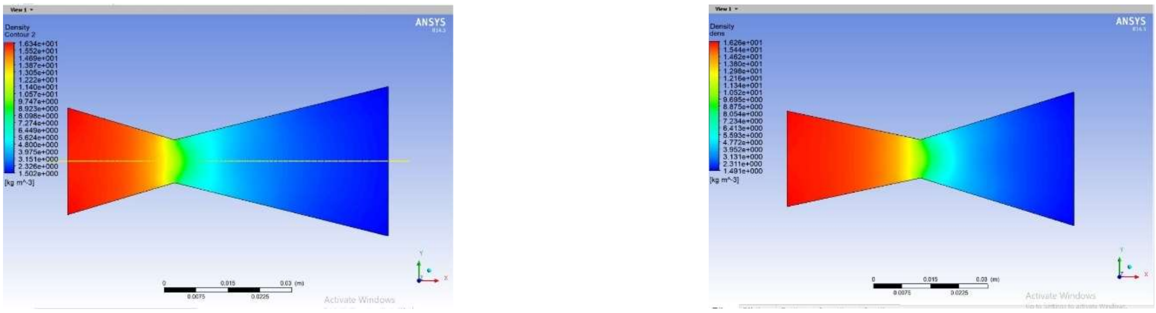
(c)Angle 14.4° **(d)Angle 15.6°**



(e)Angle 17.7°

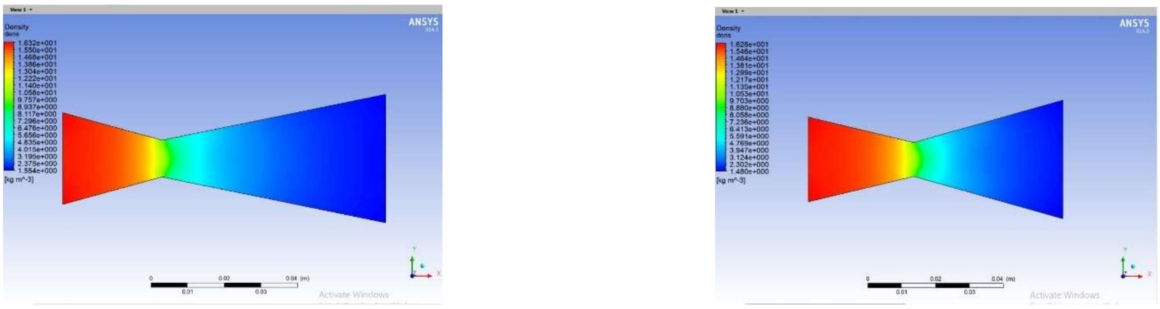
Fig No 5.13 Mach Number contours for various divergence angles at 9 bar pressure

The Density Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 9 bar.



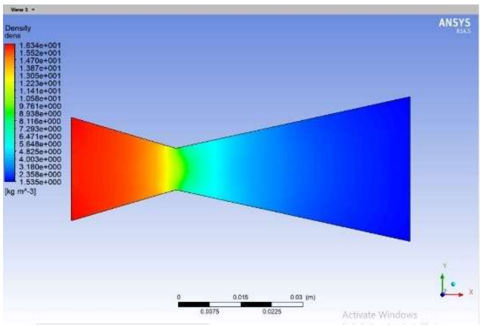
(a) Angle 11.6°

(b)Angle 12.5°



(c)Angle 14.4°

(d)Angle 15.6°

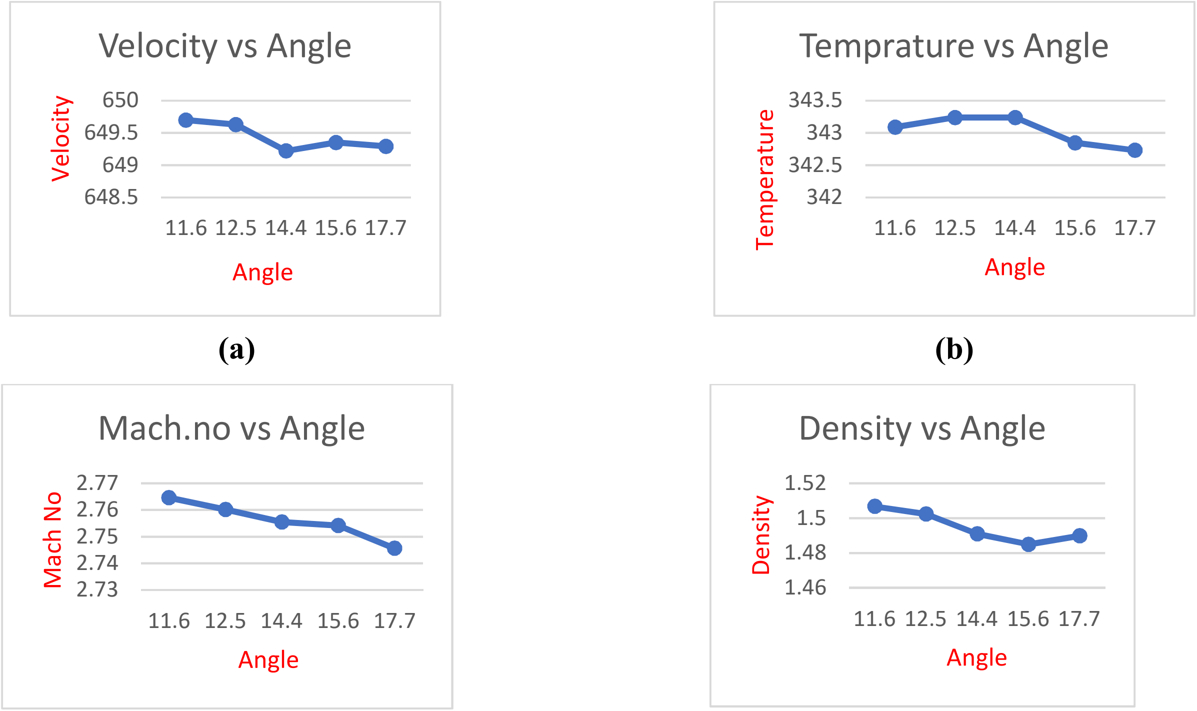


(e)Angle 17.7°

Fig No 5.14 Density contours for various divergence angles at 9 bar pressure

Table No 5.3 Comparison of divergent angles with various parameters of convergent divergent nozzle at 9 bar pressure

Angle	Inlet Pressure	Velocity	Temperature	Mach No	Density
11.6	9	649.7	343.09	2.7646	1.5067
12.5	9	649.629	343.24	2.7602	1.5023
14.4	9	649.22	343.24	2.7555	1.491
15.6	9	649.35	342.848	2.7542	1.485
17.7	9	649.29	342.73	2.7456	1.49



(a) **(b)**

(c) **(d)**

Fig 5.15 Graphs for various parameters vs divergence angles at 9 bar pressures

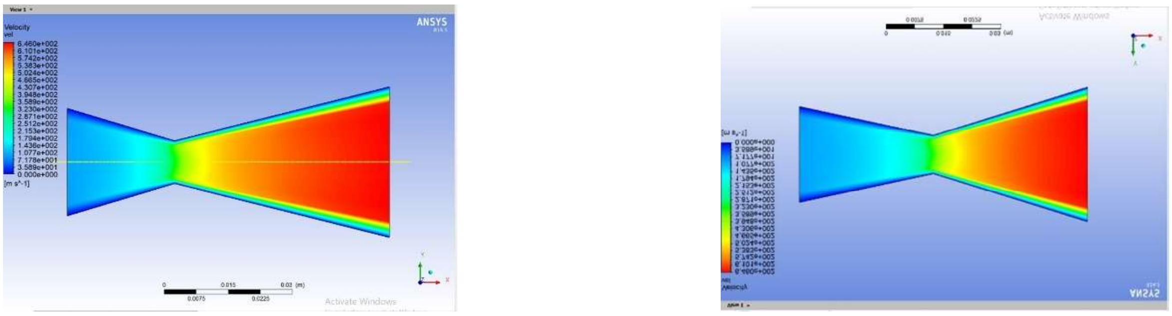
- The maximum velocity of flow through convergent divergent nozzle is obtained with the C-D Nozzle having a divergence angle of 11.6° and there is an increase in velocity obtained from the nozzles having divergence angle of 14.4° and 15.6° and the minimum velocity of flow is obtained for the nozzle having divergence angle of 17.7°.
- The Temperature is minimum at the nozzle having divergence angle of 17.7° and equal temperatures are attained for both the nozzles having divergence angle of 12.5° and 14.4° and the maximum temperature is obtained for the nozzle having divergence angle of 14.4°.
- The Mach Number is maximum for the nozzle having divergence angle of 11.6° and

minimum Mach number is for the nozzle having divergence angle of 17.7°.

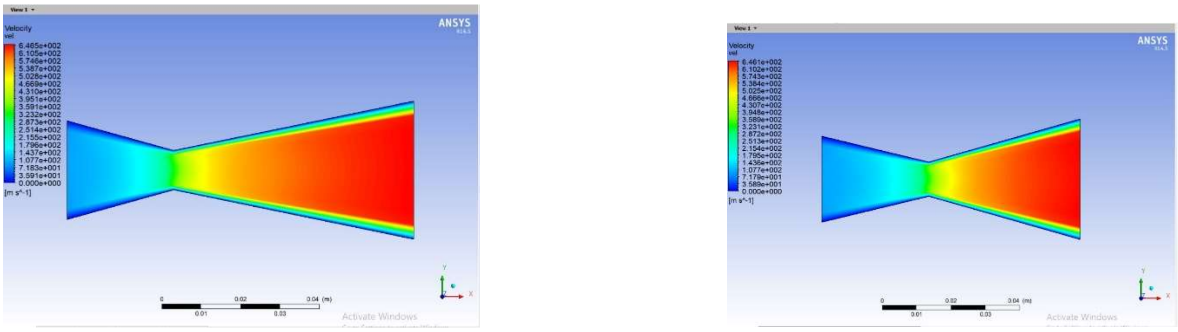
- The Density is minimum for the C-D nozzle having divergence angle 15.6° and maximum for the nozzle having the divergence angle of 11.6°.

5.1.4 Effect of divergence angle on the convergent divergent nozzle at 12 bar pressure inlet

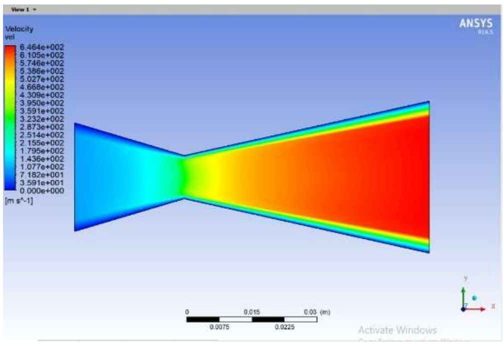
The Velocity Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 12bar.



(a) Angle 11.6° **(b)Angle 12.5°**



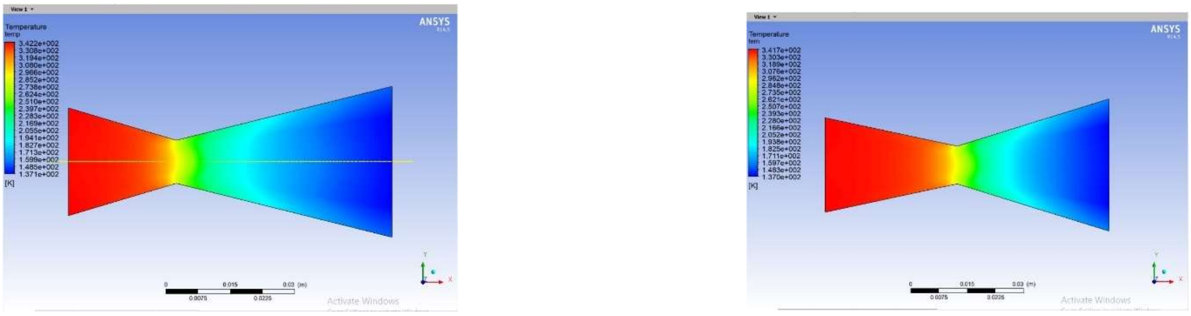
(c)Angle 14.4° **(d)Angle 15.6°**



(e)Angle 17.7°

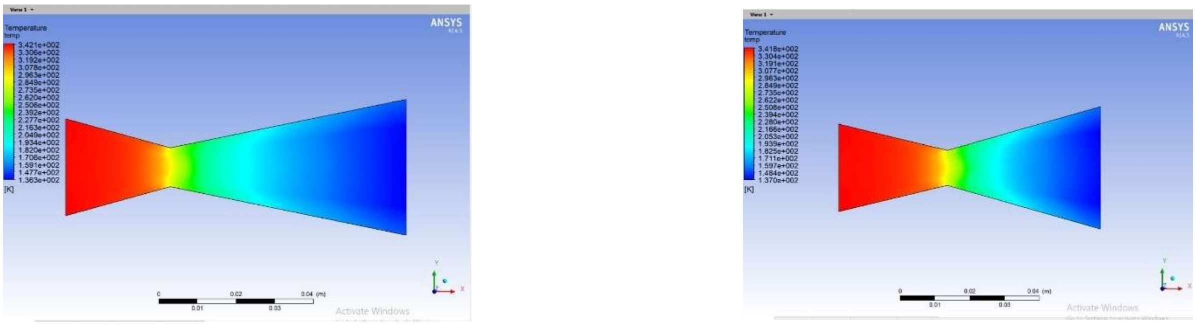
Fig No 5.16 Velocity contours for various divergence angles at 12bar pressure

The Temperature Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 12bar.



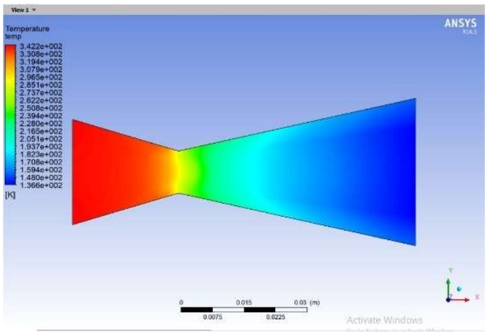
(a) Angle 11.6°

(b)Angle 12.5°



(c)Angle 14.4°

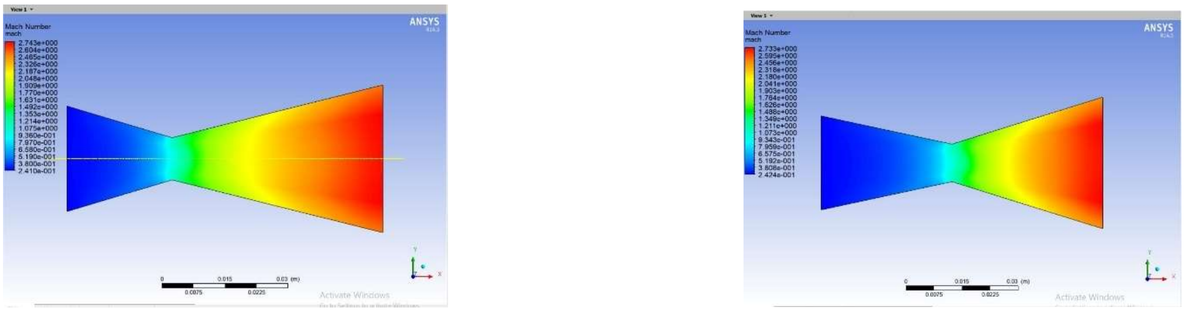
(d)Angle 15.6°



(e)Angle 17.7°

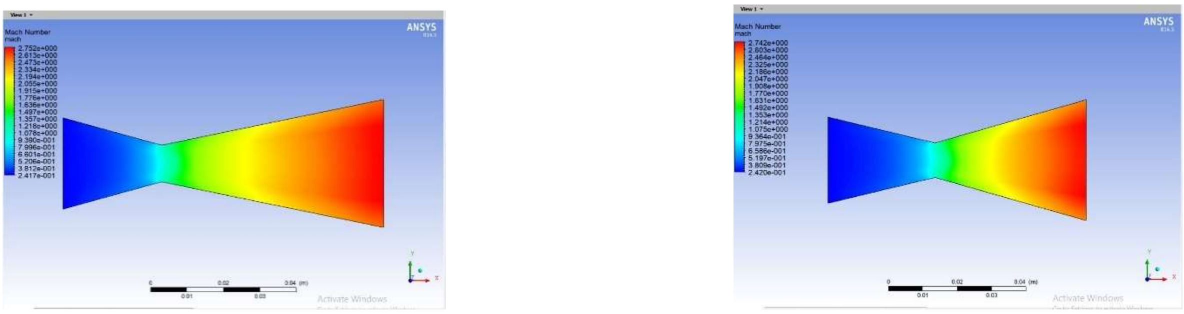
Fig No 5.17 Temperature contours for various divergence angles at 12bar pressure

The Mach Number Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 12bar.



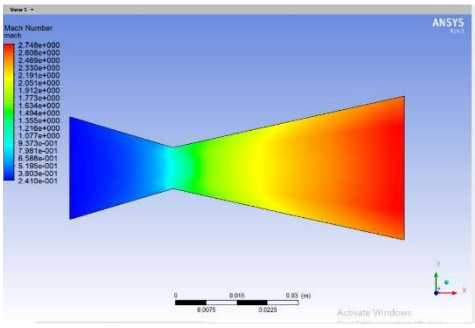
(a) Angle 11.6°

(b)Angle 12.5°



(c)Angle 14.4°

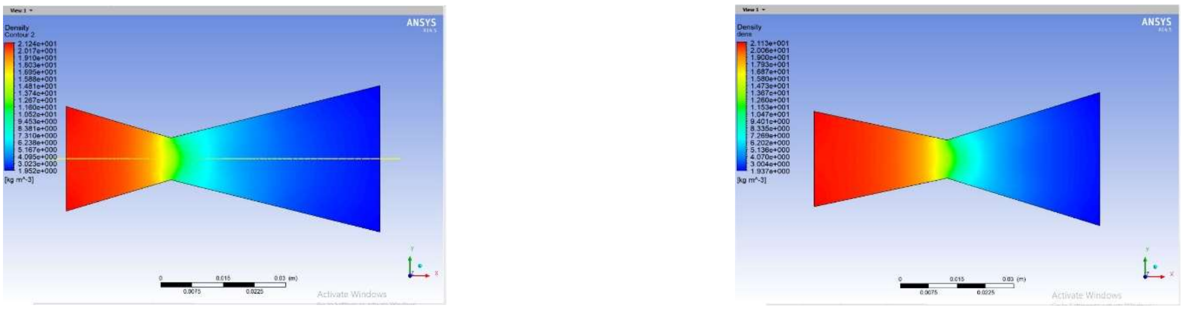
(d)Angle 15.6°



(e)Angle 17.7°

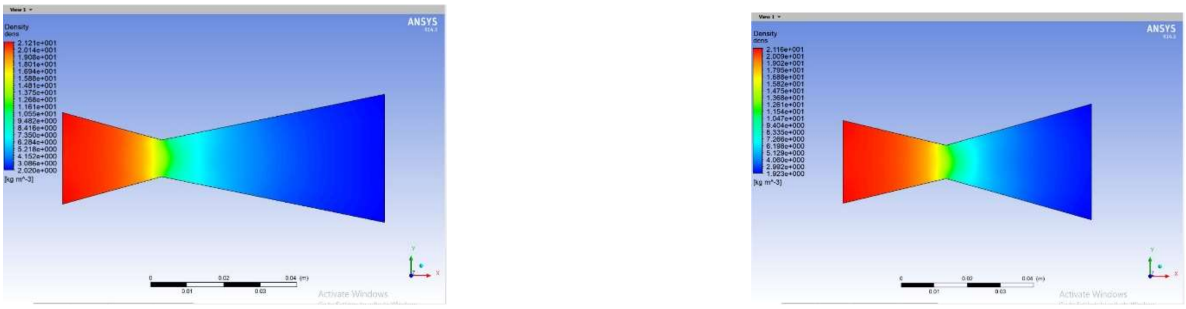
Fig No 5.18 Mach Number contours for various divergence angles at 12bar pressure

The Density Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 12bar.



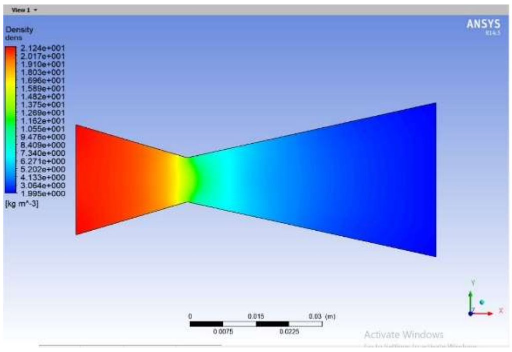
(a) Angle 11.6°

(b)Angle 12.5°



(c)Angle 14.4°

(d)Angle 15.6°

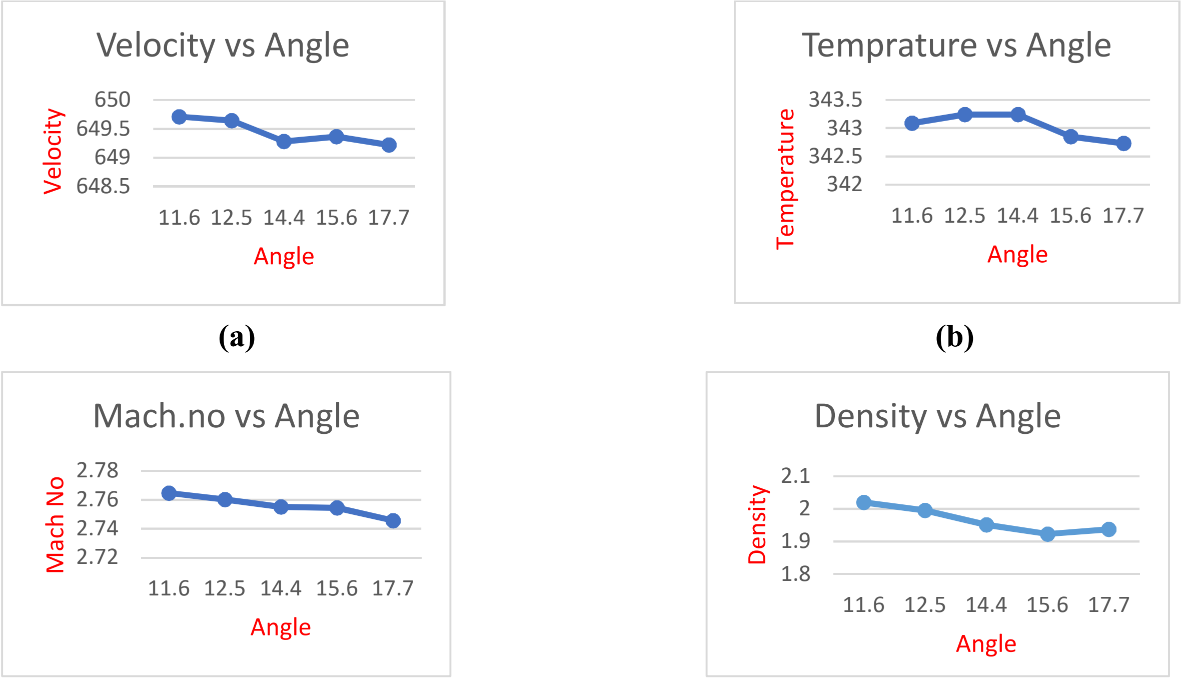


(e)Angle 17.7º

Fig No 5.19 Density contours for various divergence angles at 12bar pressure

Table No 5.4 comparison of divergent angles with various parameters of convergent divergent nozzle at 12 bar pressure

Angle	Inlet Pressure	Velocity	Temperature	Mach No	Density
11.6	12	649.71	343.09	2.76467	2.02
12.5	12	649.643	343.24	2.7602	1.995
14.4	12	649.28	343.24	2.755	1.951
15.6	12	649.366	342.848	2.7543	1.923
17.7	12	649.22	342.73	2.7456	1.9372



(a) **(b)**

(c) **(d)**

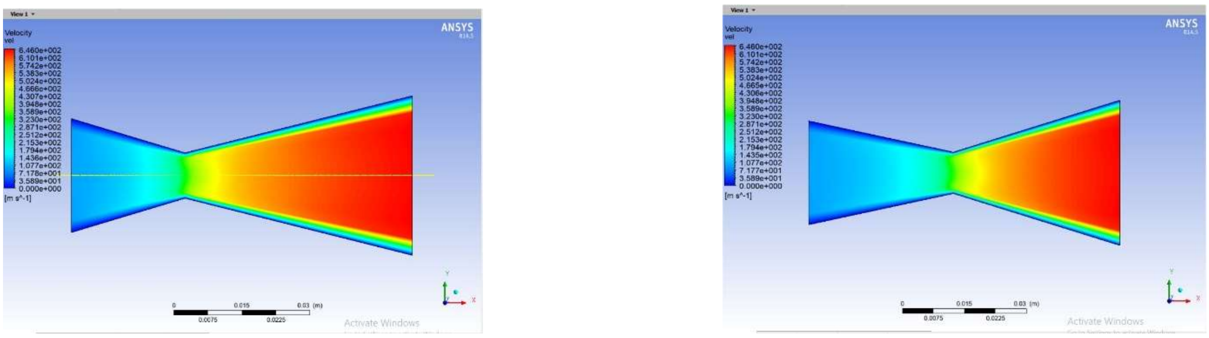
Fig 5.20 Graphs for various parameters vs divergence angles at 12 bar pressures

- The maximum velocity of flow through convergent divergent nozzle is obtained with the C-D Nozzle having a divergence angle of 11.6º and there is an increase in velocity obtained from the nozzles having divergence angle of 14.4° and 15.6° and the minimum velocity of flow is obtained for the nozzle having divergence angle of 17.7°.

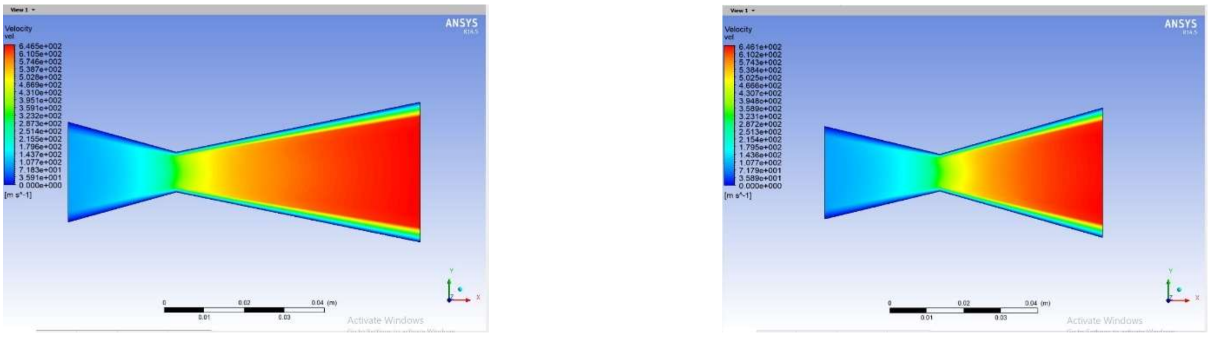
- The Temperature is minimum at the nozzle having divergence angle of 17.7° and the maximum temperature is obtained for the nozzle having divergence angle of 14.4°.
- The Mach Number is maximum for the nozzle having divergence angle of 11.6° and minimum Mach number is for the nozzle having divergence angle of 17.7°.
- The Density is minimum for the C-D nozzle having divergence angle 15.6° and maximum for the nozzle having the divergence angle of 11.6°.

5.1.5 Effect of divergence angle on the convergent divergent nozzle at 15 bar pressure inlet

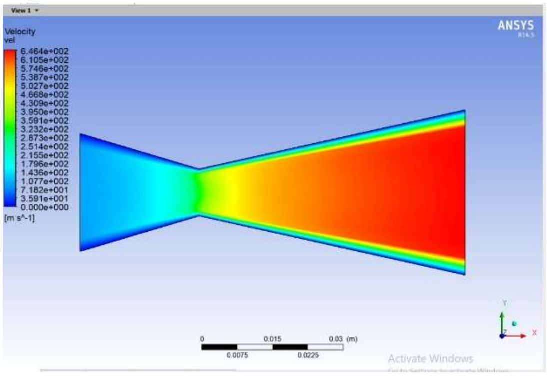
The Velocity Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 15bar.



(a) Angle 11.6° **(b)Angle 12.5°**



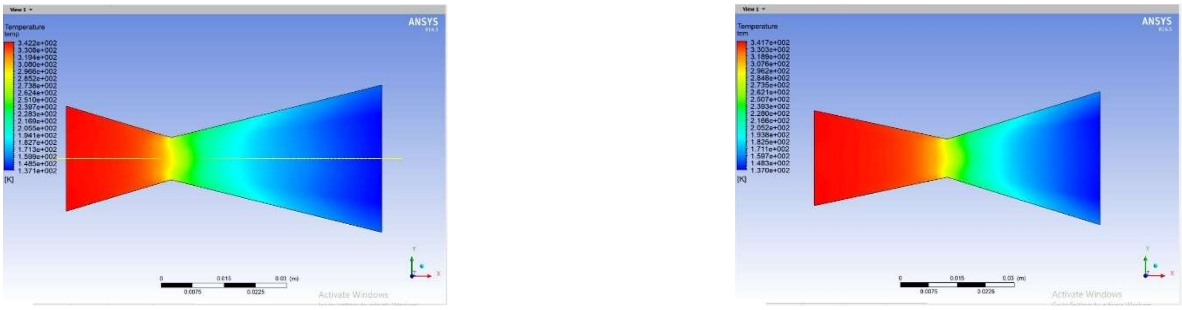
(c)Angle 14.4° **(d)Angle 15.6°**



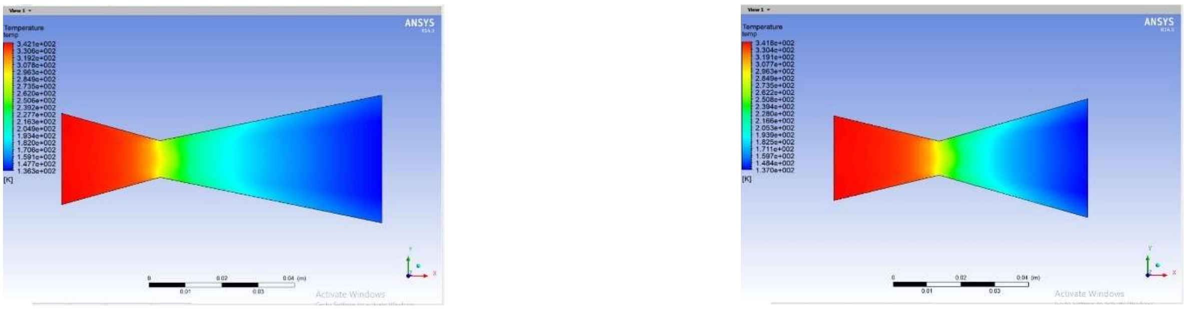
(e)Angle 17.7°

Fig No 5.21 Velocity contours for various divergence angles at 15bar pressure

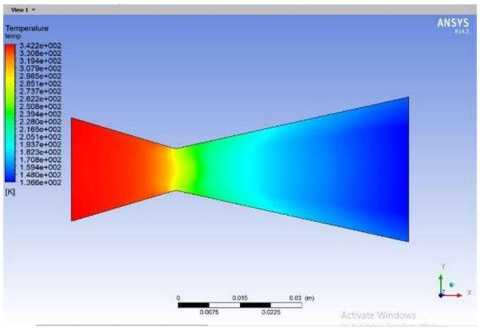
The Temperature Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 15bar.



(a) Angle 11.6° **(b)Angle 12.5°**



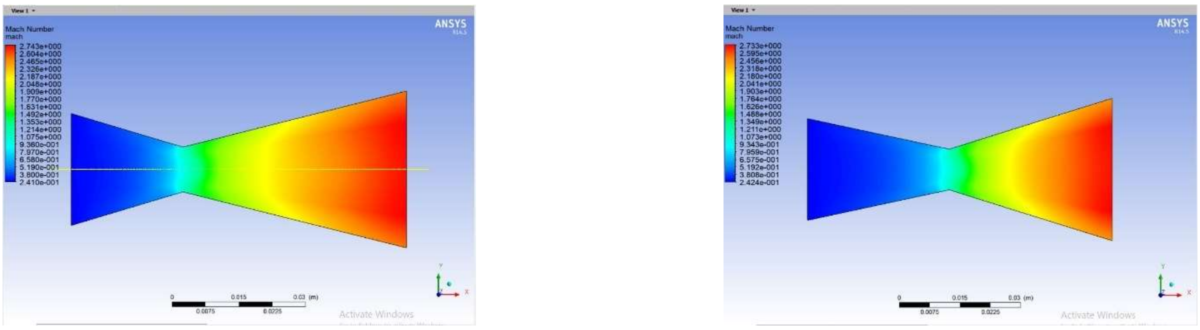
(c)Angle 14.4° **(d)Angle 15.6°**



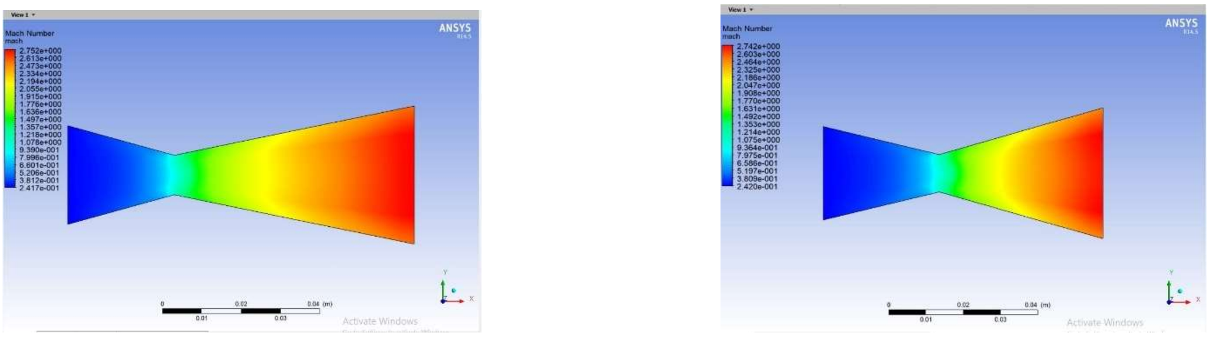
(e)Angle 17.7°

Fig No 5.22 Temperature contours for various divergence angles at 15bar pressure

The Mach number Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 15bar.

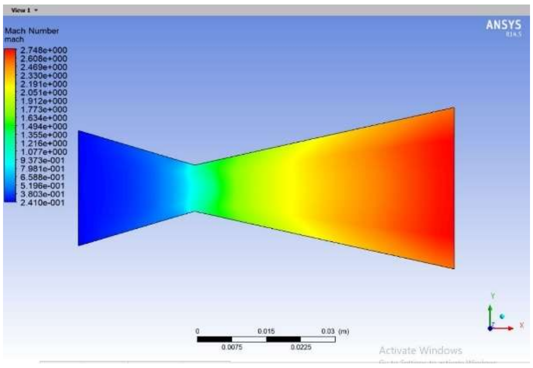


(a) Angle 11.6° **(b)Angle 12.5°**



(c)Angle 14.4°

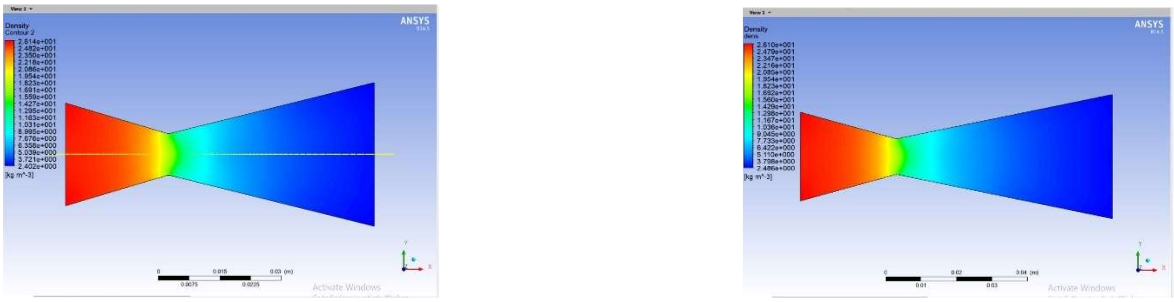
(d)Angle 15.6°



(e)Angle 17.7°

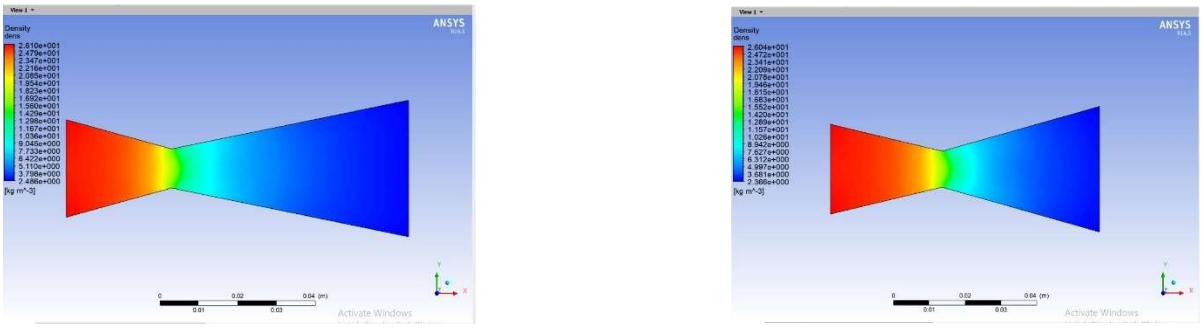
Fig No 5.23 Mach No contours for various divergence angles at 15bar pressure

The Density Contours of the different divergent angle nozzles is obtained from the FLUENT module as shown in the fig 5.5(a-b) at the input condition Pressure of 15bar.



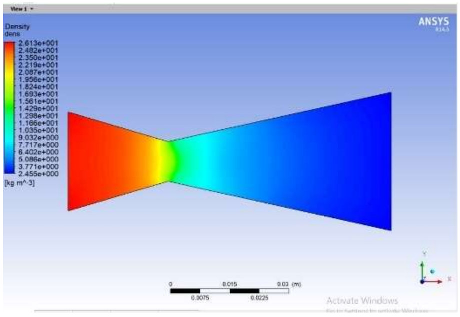
(a) Angle 11.6°

(b)Angle 12.5°



(c)Angle 14.4°

(d)Angle 15.6°

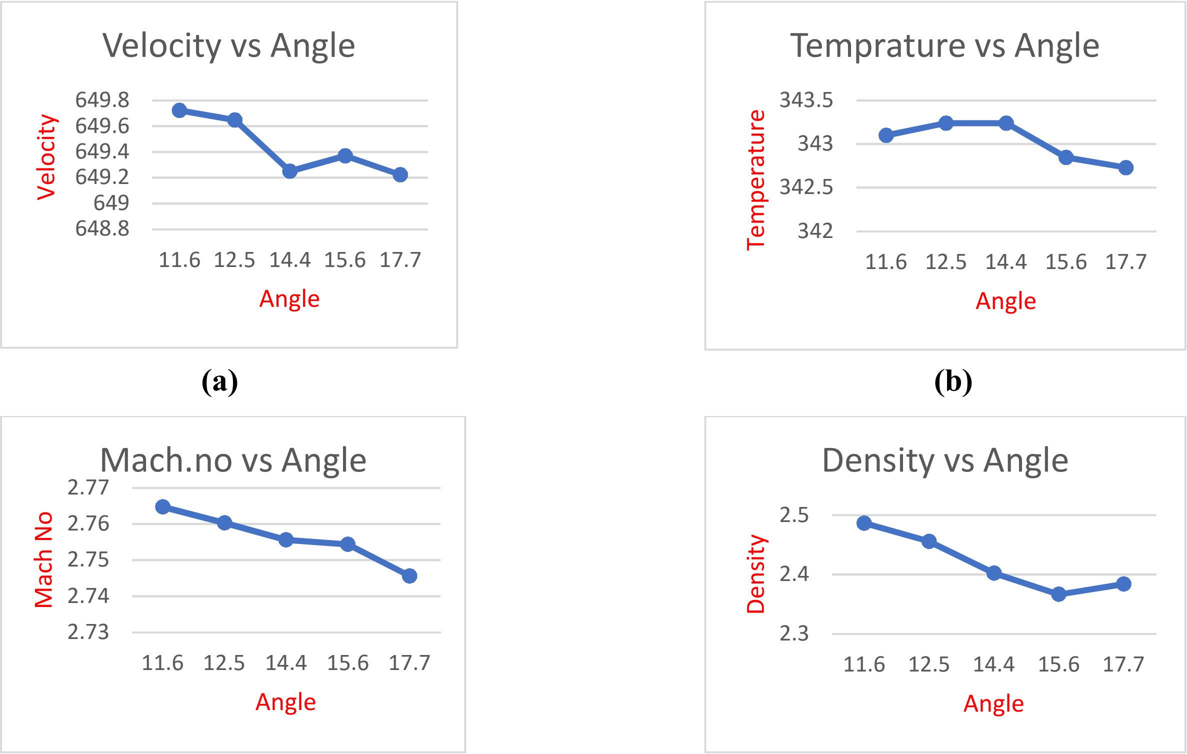


(e)Angle 17.7º

Fig No 5.24 Density contours for various divergence angles at 15bar pressure

Table No 5.5 Comparison of Divergent angles with various parameters of CONVERGENT DIVERGENT nozzle at 15 Bar Pressure

Angle	Inlet Pressure	Velocity	Temperature	Mach No	Density
11.6	15	649.723	343.099	2.76473	2.48641
12.5	15	649.649	343.24	2.76031	2.45539
14.4	15	649.25	343.24	2.7556	2.402
15.6	15	649.37	342.848	2.75432	2.36645
17.7	15	649.223	342.73	2.7456	2.3838



(a)

(b)

(c)

(d)

Fig 5.25 Graphs for various parameters vs divergence angles at 15 bar pressures

- The maximum velocity of flow through convergent divergent nozzle is obtained with the C-D Nozzle having a divergence angle of 11.6° and there is an increase in velocity obtained from the nozzles having divergence angle of 14.4° and 15.6° and the minimum velocity of flow is obtained for the nozzle having divergence angle of 17.7°.
- The Temperature is minimum at the nozzle having divergence angle of 17.7° and equal temperatures are attained for both the nozzles having divergence angle of 12.5° and 14.4° and the maximum temperature is obtained for the nozzle having divergence angle of 14.4°.
- The Mach Number is maximum for the nozzle having divergence angle of 11.6° and minimum Mach number is for the nozzle having divergence angle of 17.7°.
- The Density is minimum for the C-D nozzle having divergence angle 15.6° and maximum for the nozzle having the divergence angle of 11.6°.

CHAPTER VI

CONCLUSION

Computer Aided Solutions are developed using Fluent Analysis. Solutions are evaluated at different Divergent angles of the CONVERGENT DIVERGENT nozzle corresponding Velocity, Mach Number, Density, Temperature are determined. The Variation in increase in Inlet pressure with Mach Number was found to be identical from the Inlet throat. With the corresponding increase of inlet pressure corresponding velocity also increases. Variation in Density decrease with decrease in Inlet pressure. Temperature increases with increasing Inlet pressures.

As the classical one-dimensional inviscid theory does not completely reveal the complex flow features in a convergent divergent nozzle accurately. We used the code fluent to compute various flow parameters in Convergent Divergent Nozzle for different nozzle delivery pressures and angle of divergence.

Solutions are evaluated at different Divergent angles of the Convergent Divergent Nozzle corresponding to the Velocity, Mach Number, Density and Temperature.

The angle of divergence for the divergent portion is varied and its corresponding parameters are studied and convergence of flow is obtained at 554 iterations using Computational Fluid Dynamics (CFD).

From the solutions obtained it is observed that the optimized angle for the divergence in the Convergent Divergent Nozzle is $14^{o} – 15^{o}$ to obtain maximum velocity and higher propulsions for the rocket engines can be achieved.

CHAPTER VII

REFERENCES

1. Design and Numerical Simulation of Convergent Divergent Nozzle Ramji , Mukesh R, Inamul Hasan, ISSN: 1662-7482, Vol. 852, pp 617-624,2016.
2. Design and cfd analysis of convergent and divergent nozzle p.vinod kumar , b.kishore kumar International journal of professional engineering studies volume 9 /issue 2 / Sep 2017.
3. CFD analysis and parameter optimization of Divergent Convergent Nozzle, (Volume 3, Issue 10)-IJARND
4. Theoretical & cfd analysis of de laval nozzle nikhil d. Deshpande, Suyash s. Vidwans, Pratik r. Mahale, Rutuja s. Joshi, K. R. Jagtap International Journal of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 2, Issue- 4, April-2014.
5. Effect of Exit Diameter on the Performance of Converging – Diverging Annular Nozzle Using CFD Arjun Kundu , Devyanshu Prasad , Sarfraj Ahmed, ISSN(Online): 2319-8753 Vol. 5, Issue 6, June 2016
6. Modal Analysis of Supersonic Flow Separation in Nozzles by RAGNAR LÁRUSSON ISBN 978-91-7597-542-9
7. Farley John M, Campbell Carl E, [1960], Performance of several characteristics exhausts nozzles, NASA Lewis Research center.
8. Angelino, G., Oct. [1964], "Approximate Method for Plug Nozzle Design", AIAA Journal, Vol. 2, No. 10, pp. 1834- 1835.
9. J.Reid, [1964], An Experiment on Aerodynamics nozzles at Mach 2, Aeronautical research council reports & memoranda, R & M. No 3382.
10. G T Galesworthy, JB Robert and C Overy [1966], the Performance of conical convergent divergent nozzles of area ratio 2.44 and 2.14 in external flow. Aeronautical Research Council CP No 893.
11. TV Nguyen and JL Pieper [1996], Nozzle separation prediction techniques and controlling techniques, AIAA paper.
12. Gerald Hagemann, [1998], Advanced Rocket Nozzles, Journal of Propulsion & Power, Vol.14, and No.5.
13. Dale, D., Kozak, J., Patru, D., [2006], "Design and Testing of a Small Scale Rocket for Pico-Satellite Launching", Rochester Institute of Technology METEOR Project, Senior Design Project 06006.
14. Nicholas J Georgiadis, Teryn W DalBello, Charles J Trefny, and Albert L. Johns,[2006],Aerodynamic Design and Analysis of high performance nozzles for Mach 4 Accelerator Vehicles.