Andhra University

5

Simulation of Convergent Divergent Rocket Nozzle using CFD Analysis

Submitted to Andhra University, Vishakhapatnam for the partial fulfillment of the requirements for the award of
The degree of
In B. Tech IV/IV (SEM – II)
By

S. Gopal	**314126520151**
S. Ravi Srinivas	**314126520146**
Raghu Maharana	**314126520172**
B. Tirupati	**314126520181**
G. Raj Manesh	**314126520188**

Under the Esteemed guidance of
B. Geeta Chandra Sekhar (M.Tech)
Assistant Professor

PRAGNANAM BRAHMA
ANITS

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY AND SCIENCES

(Sangivalasa-531162, Bheemunipatnam (Mandal), Visakhapatnam Dist.)

2018

Andhra University

Department of Mechanical Engineering

Anil Neerukonda Institute of Technology & Sciences

Sangivalasa

PRAGNANAM BRAHMA

ANITS

2018

CERTIFICATE

This is to certify that the Project Report entitled "**Simulation of Convergent Divergent Rocket Nozzle using CFD Analysis**" has been carried out by S.Gopal(314126520151), S.Ravi Srinivas(314126520146), Raghu Maharana(314126520172), B.Tirupati(314126520181) and G.Raj Manesh(314126520188) under my guidance, in partial fulfillment of the requirements of Degree of Bachelor of Mechanical Engineering of Andhra University, Visakhapatnam

Approved By

Dr.B.Naga Raju
Head of the Department
Dept. of Mechanical Engineering
ANITS, Sangivalasa
Visakhapatnam.

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

B.G.CH.Sekhar
Guide

B. Geeta Chandhra Sekhar
Assistant Professor
Dept. of Mechanical Engineering
ANITS, Sangivalasa
Visakhapatnam.

THIS PROJECT IS APROOVED BY

INTERNAL EXAMINER:

Dr. B. Naga Raju
M.Tech,M.E.,Ph.d
Professor & HOD
Dept of Mechanical Engineering
ANITS, Sangivalasa,
Visakhapatnam-531 162.

EXTERNAL EXAMINER:

ACKNOWLEDGEMENTS

We express immensely our deep sense of gratitude to **Sri. B. Geeta Chandhra Sekhar,** 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 **Prof.T.V.Hanumantha rao**, Principal and **Prof.B.Naga Raju**, Head of the Department, Mechanical Engineerir.g Department, Anil Neerukonda Institute of Technology & Sciences 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 the least, we like to convey our thanks to all who have contributed either directly or indirectly for the completion of our work.

S. Gopal(314126520151)
S. Ravi Srinivas(314126520146)
Raghu Maharana(314126520172)
B. Tirupati(314126520181)
G. Raj Manesh(314126520188)

ABSTRACT

A rocket nozzle is a mechanical device which is designed to control the rate of flow, speed, direction and pressure of stream that exhaust through it. There are various types of rocket nozzles which are used depending upon the mission of the rocket. This project contains analysis over a convergent divergent rocket nozzle which is performed by varying the divergent angle. Also the various contours of nozzle like Static Pressure, Velocity, Mach Number, and Total Temperature are calculated at each type of mesh using CFD analysis software ANSYS Fluent.

Keywords:

ANSYS Fluent, Convergent, Mesh

Contents

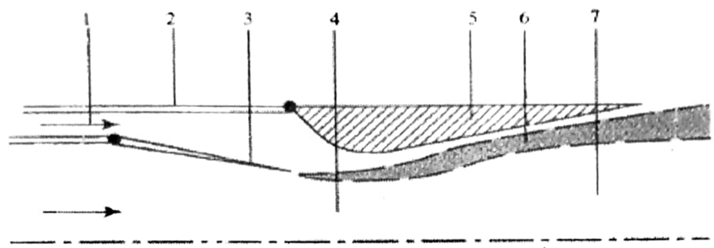
1 INTRODUCTION

1.1 NOZZLE

The nozzle may be thought of as a device that converts enthalpy into kinetic energy with no moving parts. A nozzle is used to give the direction to the gases coming out of the combustion chamber. Nozzle is a tube with variable cross-sectional area. Nozzles are generally used to control the rate of flow, speed, direction, mass, shape and/or the pressure of the exhaust stream that emerges from them. The 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. The convergent nozzle is a simple convergent duct as shown in Fig.1. When the nozzle pressure ratio Pte/Po is low, the convergent nozzle is used. The convergent nozzle has generally been used in engines for subsonic aircraft.

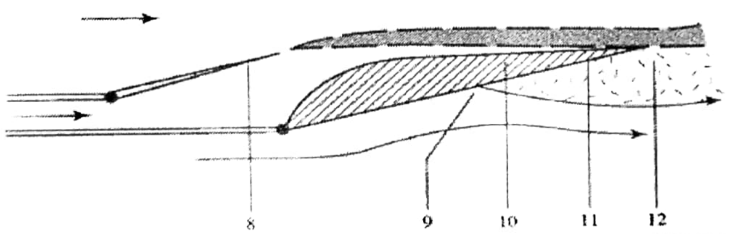
The convergent and divergent type of nozzle is known as DE-LAVALNOZZLE. Throat is the portion with minimum area in a convergent-divergent nozzle. The divergent part of the nozzle is known as nozzle exit area or nozzle exit.. In the convergent section the pressure of the exhaust gases will increase and as the hot gases expand through the diverging section attaining high velocities from continuity equation. In nozzle the combustion chamber pressure is decreased as the flow propagates towards the exit as compared to the ambient pressure i.e. pressure outside the nozzle, this result in maximum expansion known as optimum expansion and nozzle is known as adapted.

Fig 1.1 Convergent exhaust nozzle



(a) Supersonic nozzle configuration with afterburning: (1) secondary flow; (2) outer case engine, (3) movable primary nozzle shown at maximum area; (4) primary flow, effective throat; (5) movable secondary nozzle shown at maximum exit area; (6) mixing layer between primary and secondary streams; and (7) supersonic primary flow

Fig. 1.2 Supersonic Flow of Nozzle



(b) Subsonic nozzle configuration with no afterburning: (8) primary nozzle at minimum area; (9) separation point of external flow; (10) secondary nozzle at minimum area; (11) sonic primary stream; and (12) region of separated flow in external flow

Fig. 1.3 Subsonic Flow of Nozzle

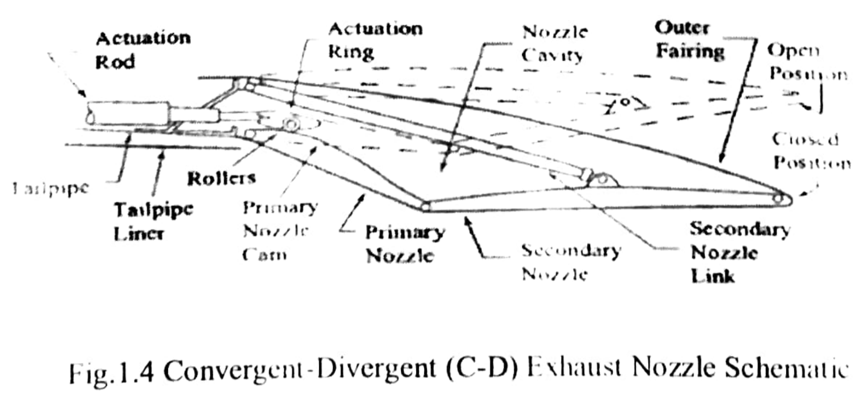
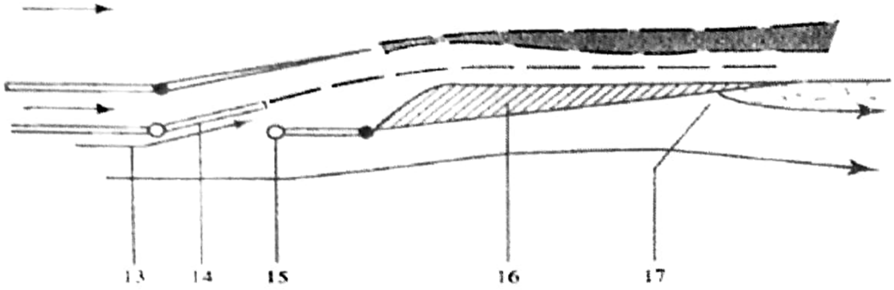


Fig.1.4 Convergent-Divergent (C-D) Exhaust Nozzle Schematic



(c) Subsonic nozzle configuration, not afterburning, and blow-in door in use: (13) tertiary flow of ambient gas into nozzle; (14) blow-in door and inflow configuration; (15) reversible hinge/latch; (16) movable secondary nozzle; and (17) separation point of external flow.

Fig. 1.5 Ejector Nozzle Configuration

The convergent-divergent nozzle is used if the nozzle pressure ratio is high. High-performance engines in supersonic aircraft generally have some form of a convergent-divergent nozzle. If the engine incorporates an afterburner, the nozzle throat is usually scheduled to leave the operating conditions of the engine upstream of the afterburner unchanged in other words, the exit nozzle area is varied so that the engine doesn't know that the afterburner is operating. Also, the exit area must be varied to match the internal and external static pressures at exit for different flow conditions in order to produce the maximum available uninstalled thrust. Earlier supersonic aircraft used ejector nozzles as shown in Fig.1.3 with their high performance turbo jets. Use of the ejector nozzle permitted by passing varying amount so inlet air around the engine, providing engine cooling, good inlet recovery, and reduced boat tail drag. Ejector nozzles can also receive air from outside the nacelle directly into the nozzle for better overall nozzle matching these are called two-stage ejector nozzles. For the modem high-performance after burning turbo fan engines, simple convergent- divergent nozzles are used without secondary air as shown in Fig. 1.4 for the F100 engine.

1.2 TYPES OF NOZZLES

1.2.1 Gas jet

A gas jet is a nozzle made for the ejection of gas or fluid in the flow stream into the surrounding environment. It is also known as fluid jet or hydro jet. These types of jets are generally present in Household equipment's like gas stoves, ovens or barbecues. In early days when there was no electricity then the gas jets were used for light. Other fluid jets are used where flow regulation is required ,like in carburettors smooth orifices are used for the regulation of the fuel flow into an engine. Another type of jet is the laminar jet. This is basically a water jet with a stream lined flow. These types of nozzles are often used in fountains.

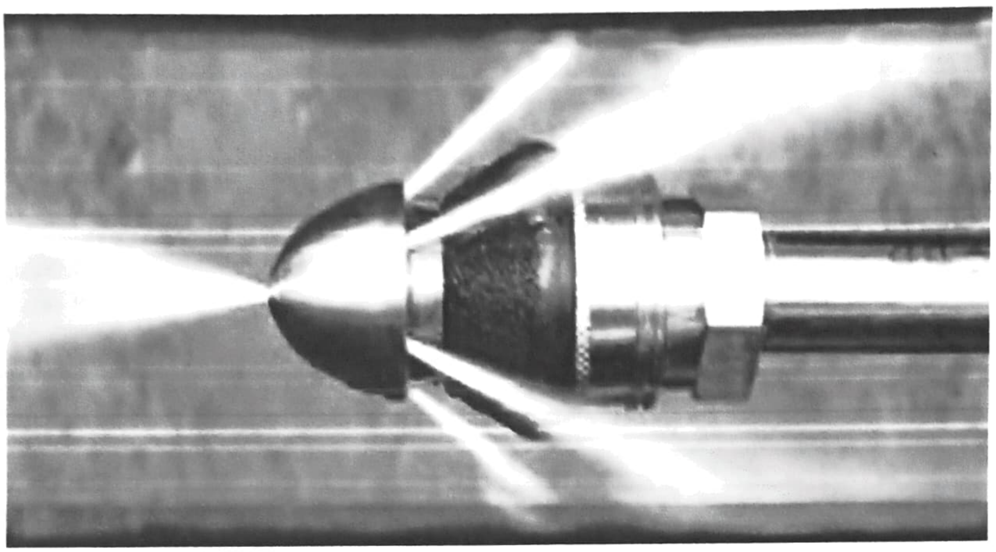


Fig 1.4 Jet nozzles

1.2.2 High velocity nozzle

The main goal is to increase the kinetic energy of the fluid at the expense of its pressure and energy. Nozzles can be defined as convergent i.e., narrowing down from a wide diameter to a smaller diameter in the direction of the flow or divergent i.e., expanding from a smaller diameter to a larger one. Convergent part of the nozzles accelerates subsonic fluids. If the pressure ratio of the nozzle is high enough to the flow will reach sonic velocity at the narrowest point i.e. the nozzle throat. This condition of the nozzle choked condition. On increasing the nozzle pressure ratio further will not increase the throat Mach number and nozzles slow fluids, if the flow is subsonic, but accelerate sonic or supersonic fluids. Convergent-divergent nozzles can therefore accelerate fluids that have choked in the convergent section to supersonic speeds. Down stream flow is free to expand to supersonic velocities. The process is more efficient than allowing a convergent nozzle to expand supersonically externally.

The shape of the divergent sectional so ensures that the direction of the escaping gases is directly backwards, as any sideways component would not contribute to thrust.

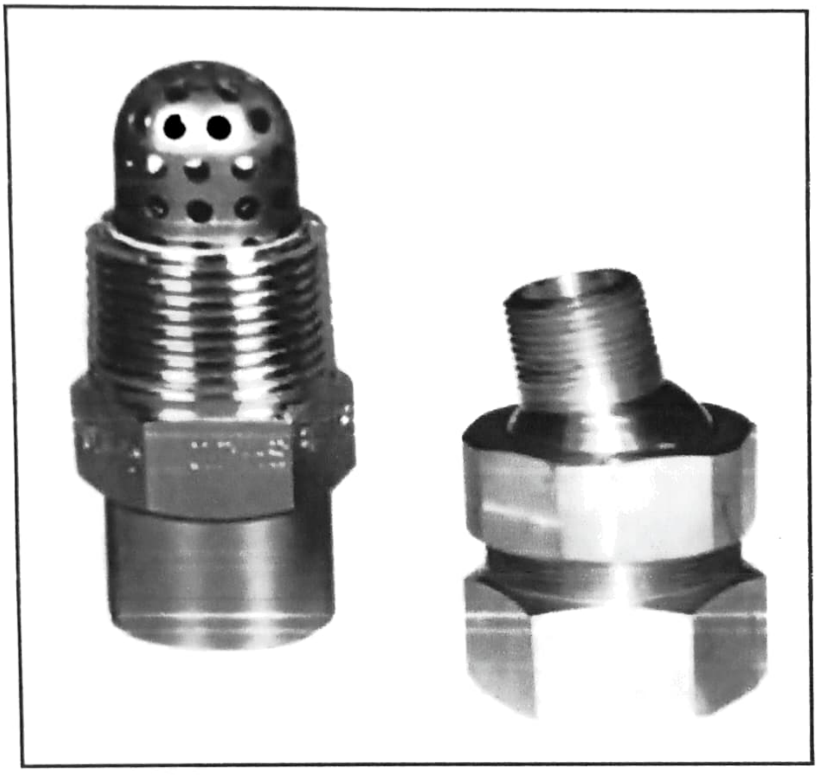


Fig. 1.6 High Velocity Nozzle

1.2.3 Propelling nozzles

A jet exhaust produces an thrust from the energy obtained from combusting fuel which is added to the inducted air. This hot air is passed through a high speed nozzle, a propelling nozzle which drastically increases its kinetic energy. For a particular mass flow, greater thrust is obtained with a higher exhaust velocity, but the best energy efficiency is obtained when the exhaust speed is well matched with the air speed. However, no jet aircraft can maintain velocity while exceeding its exhaust jet speed, due to momentum considerations. Supersonic jet engines, like those employed in fighters& commercial aircraft, need high exhaust speeds. Therefore supersonic aircraft use a convergent divergent nozzle despite weight and cost penalties. Sub sonic jet engines employ relatively low, subsonic, exhaust velocities. They thus employ simple convergent nozzles. In addition, bypass nozzles areemployedgivingevenlowerspeeds.Rocketmotorsuseconvergent-divergentnozzles with very large area ratios so as to maximize thrust and exhaust velocity and thus extremely high nozzle pressure ratios are employed. Mass flow is at a premium since all the propulsive mass is carried with vehicle, and very high exhaust speeds are desirable.

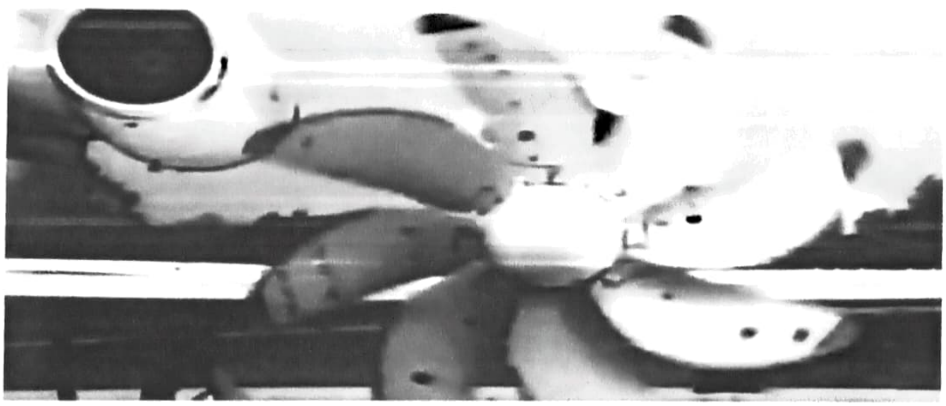


Fig. 1.7Propelling nozzle

1.2.4 Magnetic nozzles

Magnetic nozzles have also been proposed for some types of propulsion in which the flow of plasma is directed by magnetic fields instead of walls made of solid matter.

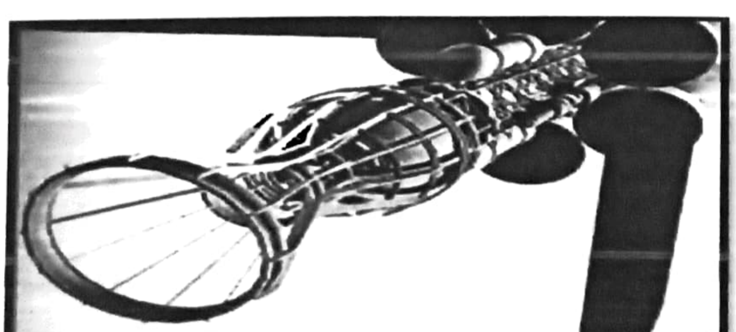


Fig 1.8 Magnetic nozzle

1.2.5 Spray nozzles

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, anti per spirants and many other 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 hand lines. Swirl nozzles inject the liquid in tangentially, and it spirals into the centre and then exits through the central hole. Due to the vortex this causes the spray to come out in a cone shape.

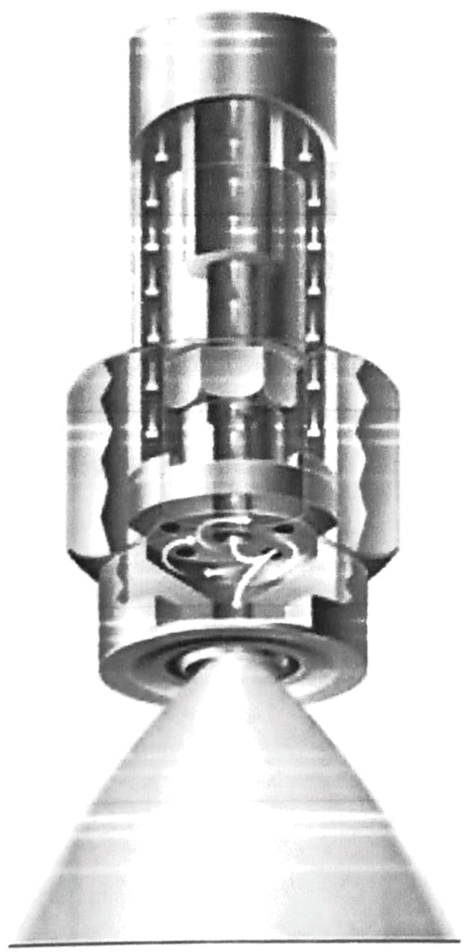


Fig 1.9 Spray nozzle

1.2.6 Vacuum nozzles

Vacuum nozzles come in several different shapes. Vacuum Nozzles are used in vacuum cleaners.

Fig 1.10 Vacuum nozzle

1.2.7 Shaping Nozzles

Some nozzles are shaped to produce a stream that is of a particular shape. For example extrusion moulding is away of producing lengths of metals or plastics or other materials with a particular cross-section. This nozzle is typically referred to as a die.

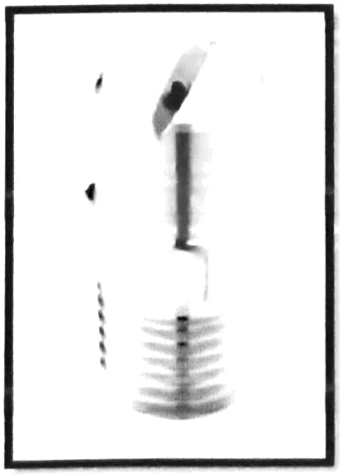


Fig 1.11 Shaping nozzle

1.3 FUNCTIONS

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_o).

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 turbo fan 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 axis symmetric 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 .

2 LITERATURE REVIEW

Before going with the project a brief study on papers related to computational fluid dynamics relating to rocket nozzle was done. Many authors gave different ideas related to their works on Computational Fluid Dynamics. The different papers reviewed are listed below:

Paradhasaradhi Natta et al. [1] In this paper the rocket nozzle is designed for Mach number 3. The mathematical model comprises of differential equations and relevant parameters that govern the behaviour of the physical system. Flow is dealt with energy equation and material as air and the density as ideal gas. Once every parameter is described, the iteration is performed till the value gets converged. We take the plotted graphs of position on one axis and pressure, temperature, velocity variations on the other axis and the results are evaluated basing on the graph.

K.M.Pandey, S.K.Yadav [2] In this paper CFD analysis of pressure and temperature for a rocket nozzle with four inlets at Mach number 2.1 is analyzed using fluent software in Ansys. The objective of the work is to simulate supersonic flow thorough rocket nozzle with combustion chamber. In this paper meshing is done using Gambit software. a numerical method is adopted to approximate governing equations along with relevant boundary conditions. This CFD numerical experiment gives the detailed physical difference between laminar and turbulent flows in the rocket nozzle.

B.V.V.Naga Sudhakar et al.[3]In this paper, it aims to study the behaviour of flow in convergent-divergent nozzle by analyzing various parameters like pressure, velocity, temperature, turbulent intensity using computational fluid dynamics (CFD). The following steps hav3e been performed to estimate the various pressures and temperatures (a) Modelling (b) Meshing (c) Pre-processing (d) Solution (e)Post-processing. Pressure, temperature, velocity and turbulence intensity contours are plotted against position (m) on graphs. Also velocity, temperature and pressure plots are drawn. From this the best convergent-divergent nozzle is selected from the simulated nozzles.

K.P.S.Surya Narayana, K.Sadhasiva Reddy [4] A convergent divergent nozzle is designed for attaining speeds that are greater than speed of sound. In this the geometry of the nozzle was created using ANSYS WORKBENCH. The meshing method used here is automatic meshing method. The boundary conditions are given for mass flow inlet, outlet and walls. The solver used in this is ANSYS FLUENT. The solution is converged after some iterations. The minimum and maximum values of pressure, velocity, temperature and Mach number are plotted.

Gutte Rajeshwara Rao et al.[5]Convergent divergent nozzle is designed for high speeds. The nozzle was designed with some specific dimensions in this method. The meshing near the boundary of the nozzle is more refined when compared to other regions of mesh. The boundary conditions are specified like inlet, outlet and walls. Fluent analysis is carried out for nozzle at different mach numbers and at different nozzle pressures. Pressure, velocity, maximum flow rate and forces are determined for this method.

Nikhil D. Deshpande et al.[6] A De-Laval nozzle is a convergent divergent nozzle, generally employed to provide supersonic jet velocity at exit of nozzle. In this method, computer simulation of nozzle is done by modelling, meshing, pre-processing. In modelling, the 2-D nozzle was done using CATIA-V5. Then meshing was created of trigonal elements and boundary conditions were given. Pre-processing was done in ANSYS FLUENT. Velocity, Temperature, pressure contours are plotted as results.

Dr.Y.V.Hanumantha Rao [7] A nozzle is used to give the direction to the gases coming out of the combustion chamber. Nozzle is a tube with variable cross sectional area. Nozzles are generally used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the exhaust stream that emerges from them. The 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 low temperature. Our study is carried using software's like gambit 2.4for designing of the nozzle and fluent 6.3.2 for analyzing the flows in the nozzle. Numerical study has been conducted to understand the air flows in a conical nozzle at different divergence degrees of angle using two-dimensional axis symmetric models, which solves the governing equations by a control volume method. The nozzle geometry co-ordinates are taken by using of method of `characteristics which usually designed for De-Laval nozzle.

Biju Kuttan PM Sajesh.[8] The CFD analysis of a rocket engine nozzle has been conducted to understand the phenomena of supersonic flow through it at various divergent angles. A two dimensional axis-symmetric model is used for the analysis and the governing equations were solved using the finite-volume method in ANSYS FLUENT software. The inlet boundary conditions were specified according to the available experimental information. The variations in the parameters like the Mach number, static pressure, turbulent intensity are being analyzed. The phenomena of oblique shock are visualized and the travel of shock with divergence angle is visualized and it was found that at 15° the shock is completely eliminated from the nozzle. Also the Mach number is found have an increasing trend with increase in divergent angle thereby obtaining an optimal divergent angle which would eliminate the instabilities due to the shock and satisfy the thrust requirements for the rocket.

Prosun Roy, Abhijit Mondal, Biswanath Barai.[9] The main objective is to analyze a rocket engine nozzle to understand the phenomena of various design conditions under different convergent angle, divergent angle and throat radius by Computational Fluid Dynamic (CFD). There have also mentioned about inlet boundary conditions with specification according to the experimental information. The paper also addresses static pressure optimization and Mach number optimization. The values on the basis of results along by optimal values of nozzle design parameters obtained from optimization techniques of Taguchi Design. Convergent angle, Divergent angle and Throat radius are considered. Also response of static pressure and Mach number values of CFD analysis in two types of inlet pressure value applied for optimal parameters of nozzle attained.

3 INTRODUCTION TO SOFTWARE PACKAGE

3.1 INTRODUCTION TO ANSYS

ANSYS is a general-purpose finite element modelling package for numerically solving a wide variety of mechanical problems. These problems include static/dynamic, structural analysis (both linear and nonlinear),heat transfer, and fluid problems, as well as acoustic and electromagnetic problems.

Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

ANSYS is the standard FEA teaching too lin Mechanical Engineering Department also used in Civil and Electrical Engineering, as well as in the Physics and Chemistry departments. ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping. With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables a reduction in the level of risk, and in the cost of ineffective designs. The multifaceted nature of ANSYS also provides a means to ensure that users are able to see the effect of a design on the whole behaviour of the product, be it electromagnetic, thermal, mechanical etc.

3.1.1 Generic Steps for Solving Any Problem in ANSYS

Like solving any problem analytically, we need to define our solution domain, physical model, boundary conditions and the physical properties in ANSYS. You then solve the problem and present the results, Compare to numerical methods, the main difference is an extra step called mesh generation. This is the step that

- Build Geometry
- Define Material Properties
- Generate Mesh
- Apply Loads, and boundary conditions
- Obtain Solution
- Present the Results

Build Geometry: In this stage construct a two or three dimensional representation of the object to be modelled and tested using the work plane coordinate system within ANSYS.

Define Material Properties: Now that the part exists, define a library of the necessary materials and material properties that compose the object (or project) being modelled. This includes thermal and mechanical properties of the object.

Generate Mesh: At this point ANSYS understands the makeup of the part. Now define how the modelled system should be broken down into finite pieces.

Apply Loads: Once the system is fully designed, the last task is to apply the system with constraints, such as physical loadings or boundary conditions.

Obtain Solution: In this step we obtain the solution. In this step we need to understand within what state (steady state, transient… etc.) the problem must be solved.

Present the Results: After the solution has been obtained, there are many ways to present ANSYS results, choose from many options such as tables, graphs, and contour plots.

3.1.2 Specific capabilities of ANSYS

- Structural Analysis
- Static Analysis
- Transient Dynamic Analysis
- Buckling Analysis
- Thermal Analysis
- Acoustics Vibration Analysis

- Coupled Fields Analysis
- Modal Analysis

3.1.2.1 Structural Analysis:

Structural analysis is the most common application of the finite element method as it implies bridges and buildings, naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools. In this we will study about the stress, strain, deformation etc. in the structure of the object.

3.1.2.2 Static Analysis:

Static Analysis is used to determine displacements, stresses, etc. under static loading conditions. ANSYS can compute both linear and nonlinear static analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep etc.

3.1.2.3 Transient Dynamic Analysis:

Transient dynamic analysis is used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned under Static Analysis above are allowed.

3.1.2.4 Buckling Analysis:

Buckling analysis is used to calculate the buckling load sand determine the buckling mode shape. Both linear (Eigen value) buckling and nonlinear buckling analysis are possible. In addition to the above analysis types, several special purpose features are available such as Fracture mechanics, Composite material analysis, Fatigue, and Beam analyses.

3.1.2.5 Thermal Analysis:

ANSYS is capable of both steady state and transient analysis of any solid with thermal boundary conditions. Steady-state thermal analysis calculate the effects of steady thermal loads on a system or component. Users often perform steady-state analysis before doing transient thermal analysis, to help establish initial conditions. Steady-state analysis also can be the last step of transient thermal analysis performed after all transient effects have diminished. ANSYS can be used to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time that are Conduction, Convection, Radiation, Heat flow rates, Heat fluxes(heat flow per unit area), Heat generation rates (heat flow per unit volume), Constant temperature boundaries etc.

A steady-state thermal analysis may be either linear, with constant material properties or nonlinear, with material properties that depend on temperature. The thermal properties of most material vary with temperature. This temperature dependency being appreciable, the analysis becomes nonlinear. Radiation boundary conditions also make the analysis nonlinear. Transient calculations are time dependent and ANSYS can both solve distributions as well as create video for time incremental displays of models.

3.1.2.6 Acoustics/Vibration Analysis:

ANSYS is capable of modelling and analyzing vibrating systems in order to that vibrate in order to analyze. Acoustics is the study of the generation, propagation, absorption, and reflection of pressure waves in a fluid medium. Applications for acoustics include the following:

- Design of concert halls, where an even distribution of sound pressure is desired.
- Noise minimization in machine shops.
- Noise cancellation in automobiles.
- Under water acoustics.
- Design of speakers, speaker housings, acoustic filters, mufflers, and many other similar devices.

Within ANSYS, an acoustic analysis usually involves modelling a fluid medium and the surrounding structure. Characteristics in question include pressure distribution in the fluid at different frequencies, pressure gradient, and particle velocity, the sound pressure level, as well as, scattering, diffraction, transmission, radiation, and dispersion of acoustic waves. A coupled acoustic analysis takes the fluid-structure interaction into account. An uncoupled acoustic analysis models only the fluid and ignores any fluid-structure interaction. The ANSYS program assumes that the fluid is compressible, but allows only relatively small pressure changes with respect to the mean pressure. Also, the fluid is assumed to be non-flowing and in viscid (that is, viscosity causes no dissipative effects). Uniform mean density and mean pressure are assumed, with the pressure solution being the deviation from the mean pressure, not the absolute pressure.

3.1.2.7 Coupled Field Analysis:

A coupled-field analysis is an analysis that takes into account the interaction (coupling) between two or more disciplines (fields) of engineering. A piezoelectric analysis, for example, handles the interaction between the structural and electric fields: it solves for the voltage distribution due to applied displacements, or vice versa. Other examples of coupled-field analysis are thermal-stress analysis, thermal-electric analysis, and fluid-structure analysis. Some of the applications in which coupled-field analysis may be required are pressure vessels (thermal-stress analysis), fluid flow constrictions (fluid-structure analysis), induction heating (magnetic- thermal analysis), ultrasonic transducers (piezo electric analysis), magnetic forming (magneto-structural analysis), and micro electromechanical systems (MEMS).

3.1.2.8 Modal Analysis:

A modal analysis is typically used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It can also serve as a starting point for another, more detailed, dynamic analysis, such as a harmonic response or full transient dynamic analysis. Modal analyses, while being one of the most basic dynamic analysis types available in ANSYS. can also be more computationally time consuming than a typical static analysis. A reduced solver, utilizing automatically or manually selected master degrees of freedom is used to drastically reduce the problem size and solution time.

3.1.2.9 Harmonic Analysis:

Harmonic analysis is used extensively by companies who produce rotating machinery. ANSYS Harmonic analysis is used to predict the sustained dynamic behaviour of structures to consistent cyclic loading. Examples of rotating machines which are subjected to harmonic loading are Gas Turbines for Aircraft and Power Generation, Steam Turbines, Wind Turbine, Water Turbines, Turbo pumps, Internal Combustion engines, Electric motor sand generators, Gas and fluid pumps, Disc drives etc.

A harmonic analysis can be used to verify whether or not a machine design will successfully over come resonance, fatigue, and other harmful effects of forced vibrations.

3.2 INTRODUCTION TO CFD

Fluid (gas and liquid) flows are governed by partial differential equations which represent conservation laws for the mass, momentum and energy.

Computational Fluid Dynamics (CFD) is the art of replacing such PDE systems by a set of algebraic equations which can be solved by using digital computers.

Computational Fluid Dynamics provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of

- Mathematical modelling (partial differential equations)
- Numerical methods (discretization and solution techniques)
- Software tools (solvers, pre-and post-processing utilities)

CFD enables scientists and engineers to perform numerical experiments (i.e. computer simulations) in a virtual flow laboratory.

Computational Fluid Dynamics (CFD) is a computer based mathematical modelling tool that can be considered the amalgamation of theory and experimentation in the field of fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena. It is now widely used and is acceptable as a valid engineering tool in industry.

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reaction (e.g. combustion), and related phenomena by solving the mathematical equations that govern these processes using a numerical algorithm on a computer. The technique is very powerful and spans a wide range of industrial and non-industrial application areas.

3.2.1 Governing Equations of Fluid Flow:

The governing equations of fluid flow represent mathematical statements of the conservation laws of physics. Each individual governing equation represents a conservation principle. The fundamental equations of fluid dynamics are based on the following universal laws of conservation. They are

- Conservation of mass
- Conservation of momentum
- Conservation of energy

3.2.1.1 Continuity Equation:

The equation based on the principle of conservation of mass is called continuity equation. The conservation of mass law applied to a fluid passing through an infinitesimal, fixed control volume yields the following equation of continuity,

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \qquad (3.1)$$

$$\frac{\partial \rho}{\partial t}+div(\rho V)=0 \qquad (3.2)$$

Where 'ρ' is the fluid density, u, v, and w is the fluid velocity vectors. For an incompressible flow, the density of each fluid element remains constant.

3.2.1.2 Momentum Equation:

The equations based on the laws of conservation of momentum or on the principle of momentum, states that, the net force acting on fluid mass is equal to the change in momentum of flow per unit time in that direction. The Navier-Stokes equations in conservative form

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}+\rho f_x \qquad (3.3)$$

$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+\rho f_y \qquad (3.4)$$

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z}=-\frac{\partial p}{\partial z}+\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+\rho f_z \qquad (3.5)$$

Unsteady Convective Pressure Diffusive Source

Where (according to Newton's Law of Viscosity),

$$\tau_{xy} = \tau_{yx} = \mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right) \quad (3.6)$$

$$\tau_{xz} = \tau_{zx} = \mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right) \quad (3.7)$$

$$\tau_{yz} = \tau_{zy} = \mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right) \quad (3.8)$$

$$\tau_{xx} = \lambda(\nabla.V) + 2\mu\frac{\partial u}{\partial x}, \tau_{yy} = \lambda(\nabla.V) + 2\mu\frac{\partial v}{\partial y}, \tau_{zz} = \lambda(\nabla.V) + 2\mu\frac{\partial w}{\partial z} \quad (3.9)$$

$\lambda = \frac{-2\mu}{3}$ which is Stokes Hypothesis (3.10)

The Navier-Stokes equations form the basis upon which the entire science of viscous flow theory has been developed. In general the continuity and energy equations are also included in the Navier-Stokes equation.

3.2.1.3 Energy Equation:

This equation is based on the principle of conservation of energy; the energy equation is derived from first law of thermodynamics which states that the rate change of energy of a fluid particle is equal to the rate of heat addition to the fluid particle plus the rate of work done on particle, which is

$\rho\frac{DE}{Dt}$ = The rate of change energy of a fluid particle

E = Internal energy + kinetic energy + gravitational energy

$$E = i + \tfrac{1}{2}\left(u^2 + v^2 + w^2\right) + g \quad (3.11)$$

$-\frac{\partial q_x}{\partial x} - \frac{\partial q_y}{\partial y} - \frac{\partial q_z}{\partial z} = -div\ q$ = The rate of heat addition to the fluid particle. (3.12)

$q_x, q_y, q_z = -k\frac{\partial T}{\partial x}, -k\frac{\partial T}{\partial y}, -k\frac{\partial T}{\partial z}$ = Heat flux components (3.13)

$$\left[\frac{\partial[u(-p+\tau_{xx})]}{\partial x}+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(u\tau_{zx})}{\partial z}\right]\delta x\delta y\delta z$$ = Net rate of work done by force in 'x' direction (3.14)

Energy equation in conservative form:

$$\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\nabla\cdot\left[\rho\left(e+\frac{V^2}{2}\right)V\right]=\rho q+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}$$

$$+\frac{\partial(u\tau_{xx})}{\partial x}+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(u\tau_{zx})}{\partial z}+\frac{\partial(v\tau_{xy})}{\partial x}+\frac{\partial(v\tau_{yy})}{\partial y}+\frac{\partial(v\tau_{zy})}{\partial z}+\frac{\partial(w\tau_{xz})}{\partial x}+\frac{\partial(w\tau_{yz})}{\partial y}+\frac{\partial(w\tau_{zz})}{\partial z}+\rho f.V \tag{3.15}$$

A typical CFD simulation consists of several stages, described below.

- **Approximation of the geometry:**

  The geometry of the physical system needs to be approximated by a geometric CAD type model. The more closely the model geometry represents the actual geometry, the more accurate the results are likely to be.

- **Creation of the numerical grid within the geometrical model:**

  To identify the discrete, finite location sat which the variables are to be calculated, the geometry is divided into a finite number of cells that makeup the numerical grid. Before doing this, it is necessary to identify the physical flow phenomena expected (turbulence, compressible flow, shocks, combustion, multiphase flow, mixing, etc.) so the grid generate dis suitable to capture these phenomena.

- **Selection of models and modelling parameters:**

  Once the geometry and grid have been established, the mathematical models and parameters for those phenomena are then selected and boundary conditions defined throughout the domain.

- **Calculation of the variable values:**

  Discretization yields a large number of algebraic equations (one set for each cell). These equations are then generally solved using an iterative method, starting with a first guess value for all variables and completing a computational cycle. Error or residual values are computed from the discredited equations and the calculations repeated many times, reducing the residual values, until a sufficiently converged solution is judged to have been reached.

- **Determination of a sufficiently converged solution:**

  The final stage in the solution process is to determine when the solution has reached a sufficient level of convergence. When the sum of the residual values around the system becomes sufficiently small, the calculations are stopped and the solution is considered converged. A further check is that additional iterations produce negligible changes in the variable values.

- **Post Processing:**

  Once a converged solution has been calculated, the results can be presented as numerical values or pictures, such as velocity vectors and contours of constant values (e.g. pressure or velocity).

- **Solution Verification and Validation:**

  Once the solution process is complete, each solution should be verified and validated. If this cannot be completed successfully, re-simulation may be required, with different assumption sand/or improvements to the grid, models and boundary conditions used.

. Normally the programs are run on workstations or supercomputers. At the end, we can get our simulation results. We can compare and analyze the simulation results with experiments and the real problem. If the results are not sufficient to solve the problem, we have to repeat the process until find satisfied solution. This is the process of CFD.

3.2.2 Advantages of CFD

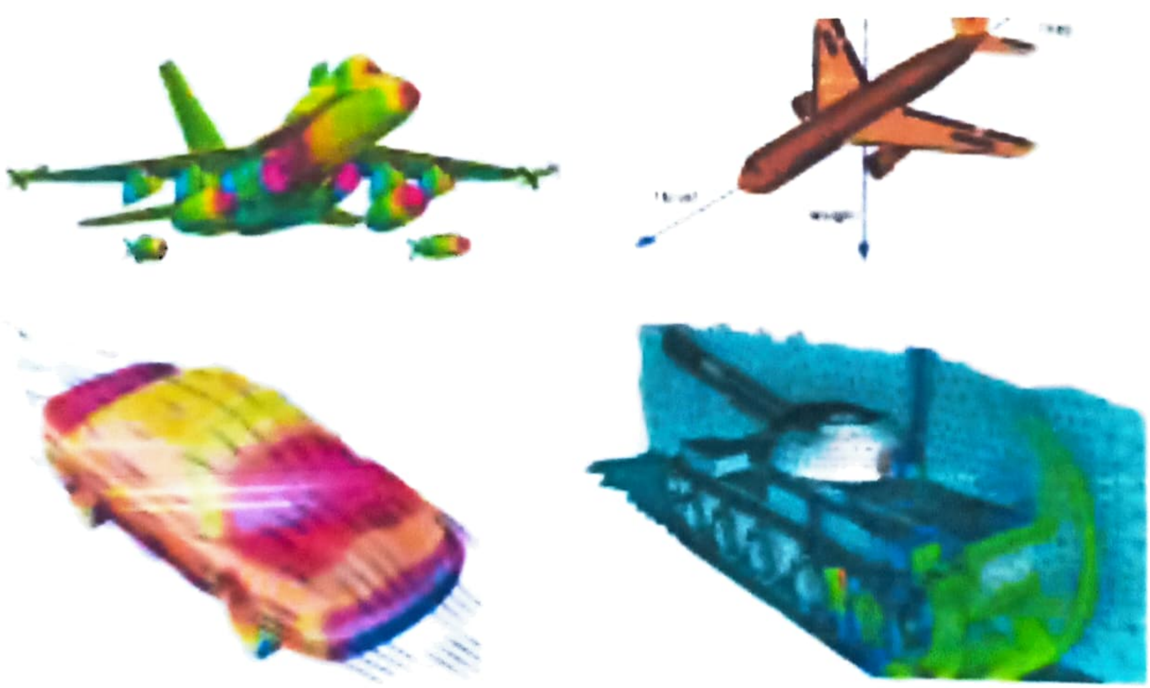
Numerical simulations of fluid flow (will) enable

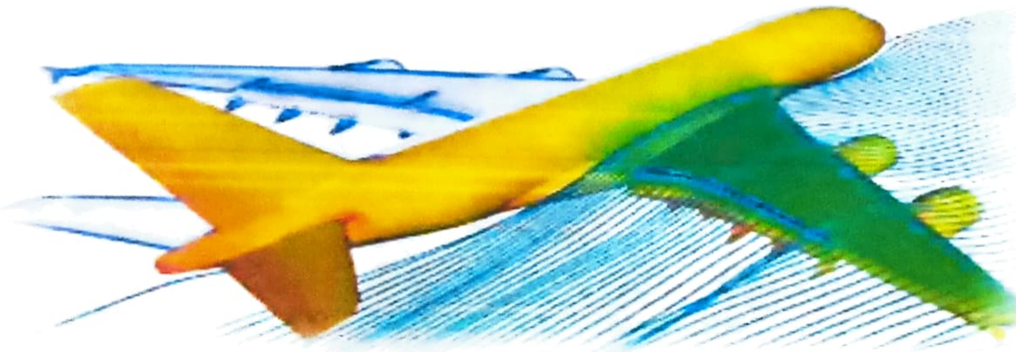
- Architects to design comfortable and safe living environments

- Designers of vehicles to improve the aero dynamic characteristics
- Chemical engineers to maximize the yield from their equipment
- Petroleum engineers to device optimal oil recovery strategies
- Surgeons to cure arterial diseases (computational hemodynamic)
- Meteorologists to forecast the weather and warn of natural disasters
- Safety experts to reduce health risks from radiation and other hazards
- Military organizations to develop weapons and estimate the damage
- CFD practitioners to make big bucks by selling colourful pictures.

3.2.3 Application of Computational Fluid Dynamics:

As CFD has so many advantages, it is already generally used in industry such as aerospace, automotive, biomedicine, chemical processing, heat ventilation air condition, hydraulics, power generation, sports and marine etc.





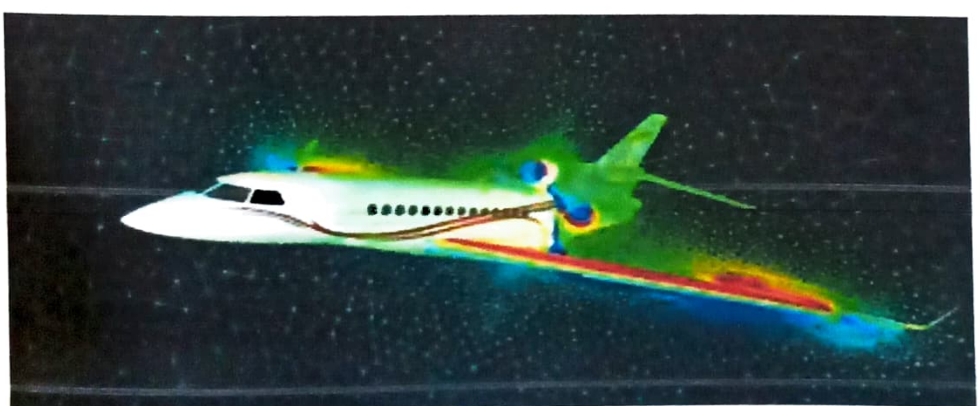


Fig 3.1 Applications of Computational Fluid Dynamics

4 MODELLING OF ROCKET NOZZLE

4.1 Modelling:

The 2-Dimensional modelling of the nozzle was done using ICEMCFD and file was saved in. jpeg format. The dimensions of the rocket nozzle are presented in the table given below.

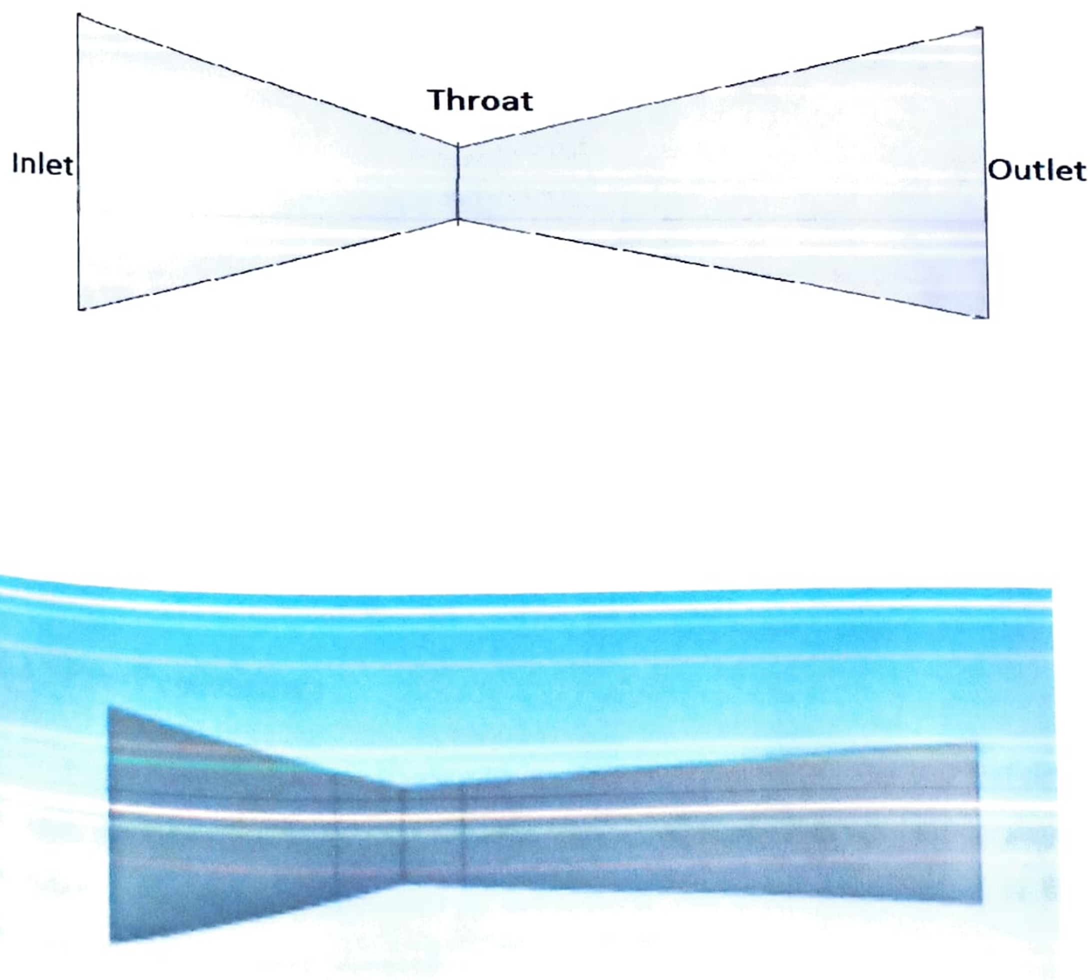


Fig4.1 Nozzle Boundary Conditions

Table 4.1 NOZZLE DIMENSIONS

PARAMETER	DIMENSIONS
Total nozzle length(mm)	75
Inlet diameter(mm)	25
Throat diameter(mm)	10
Outlet diameter(mm)	varies according to the divergent angle
Divergent angle(deg)	10,15,20,25,30,35.

4.2 Meshing:

After modelling, meshing is done using ANSYS ICEM CFD software.

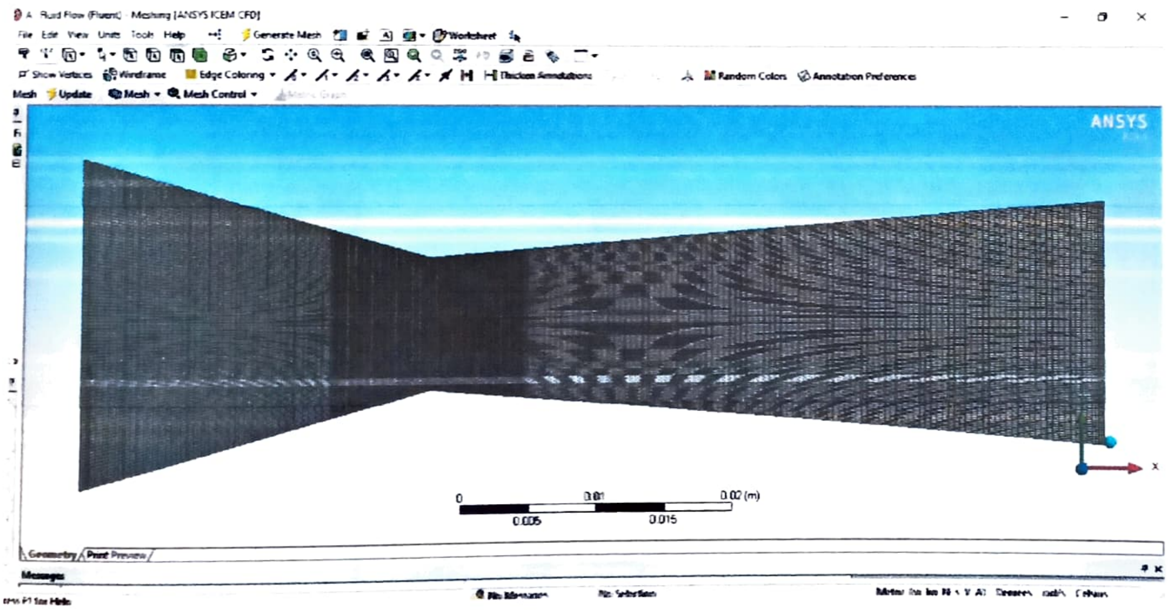


Fig 4.2 Meshed rocket nozzle

4.3 Pre-Processing:

Pre-processing of the nozzle was done in ANSYS FLUENT. 2-D and double precision settings were used while reading the mesh. The mesh was scaled since all dimensions were initially specified in millimetre. The mesh was checked in fluent and no critical errors were reported.

PROBLEMSETUP:

Table 4.2 Boundary Conditions

General	Solver type: Density-based2D Space Axi-symmetric
Models	Energy equation: On Viscous model: Laminar
Materials	Density: Ideal gas Cp: 1006.33J/Kg-Kγ: 1.19 Viscosity: Sutherland
Boundary Conditions	Inlet Pressure: 3 bar Inlet Temperature: 500K Outlet Temperature: 300K

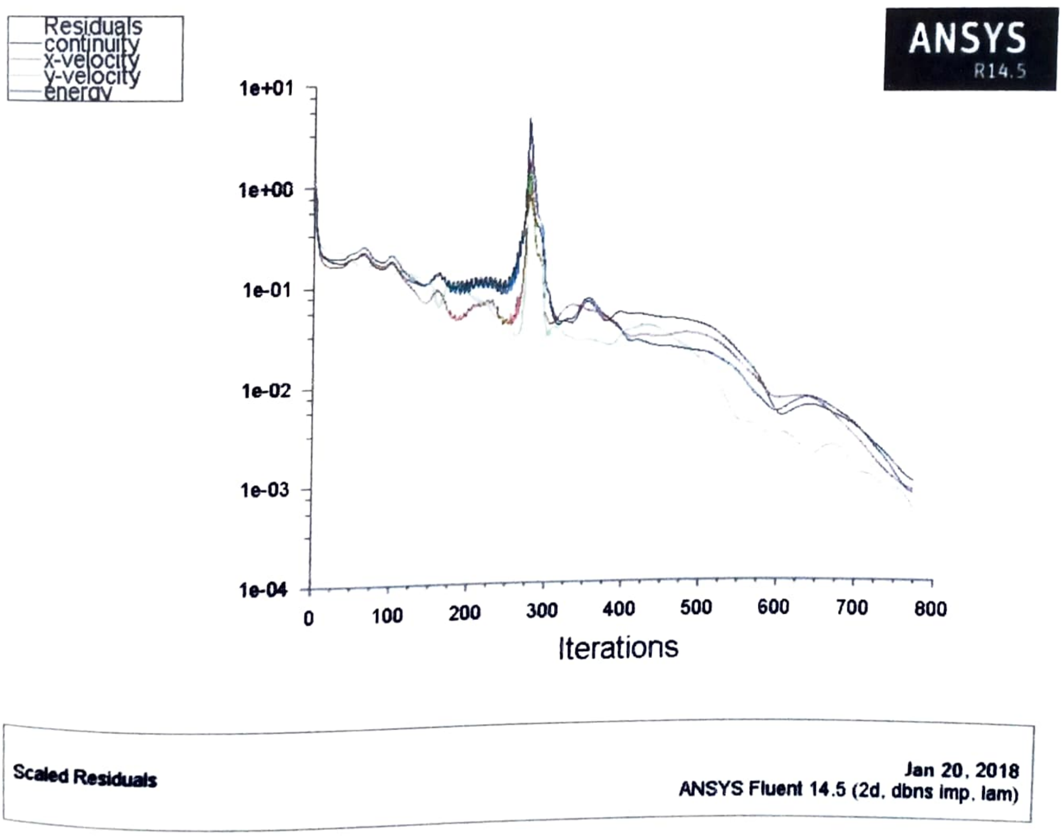


Fig 4.3 Total number of iterations are 710 in ANSYS FLUENT

4.4 Solution:

4.4.1 Mach number plot:

Graph plotted between velocity magnitude Vs position shows max velocity in convergent and divergent position, no at range 0.03-0.05. Max velocity costing at position 600 as 2.00e+00

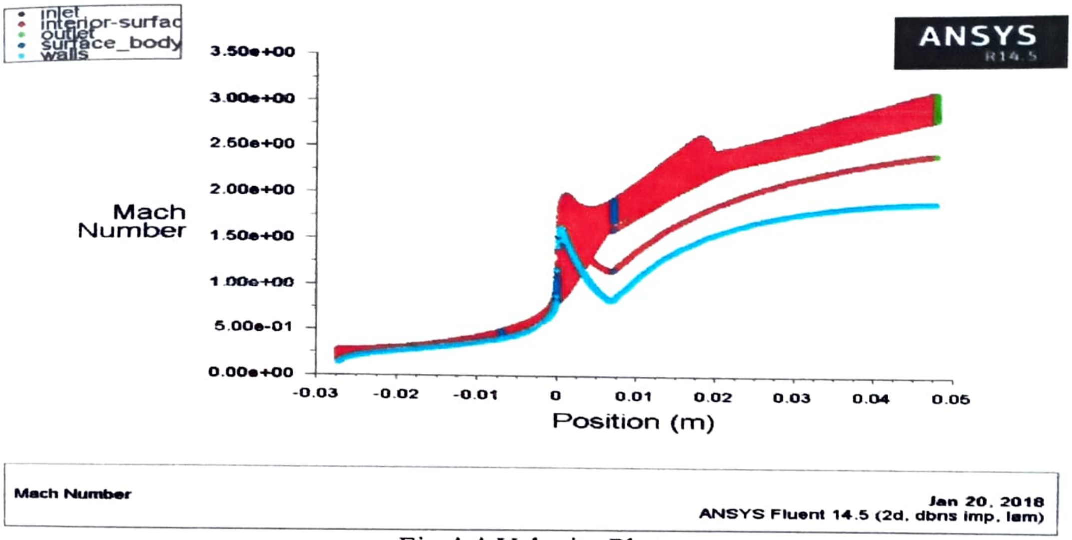


Fig 4.4 Velocity Plot

4.4.2 Static pressure plot:

Graph plotted between static pressure Vs pressure shows sudden collapse of pressure at convergent-divergent position range lost form 3.00e+05 to -5.00e+00

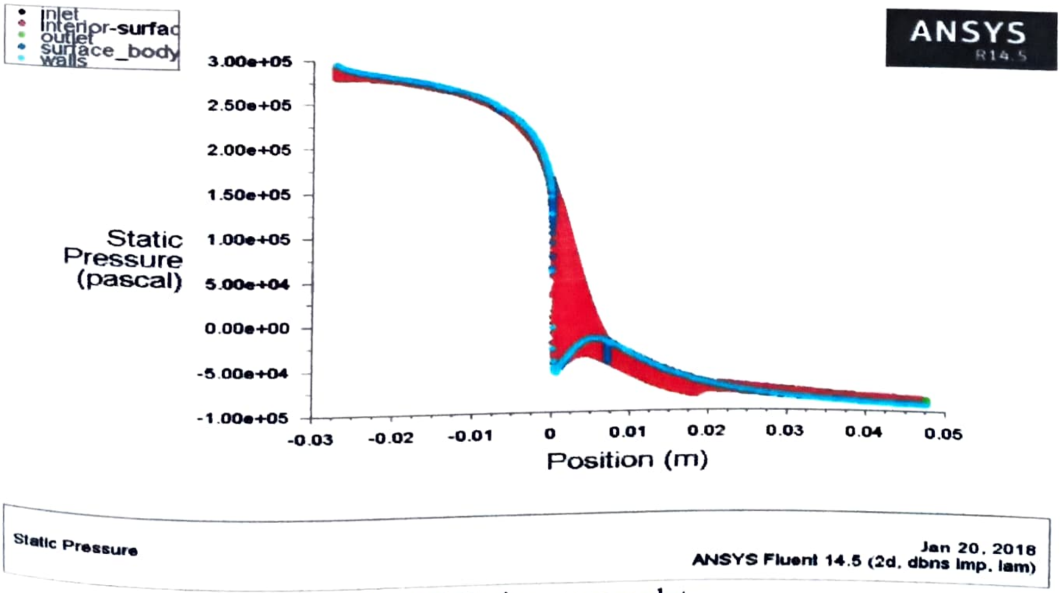


Fig 4.5 Static pressure plot

4.4.3 Static temperature plot:

Graph plotted between static temperature Vs temperature gives results max temperature recoded at before convergent space and outlet walls space. Range between 2.20e+02 to 4.00e+02.

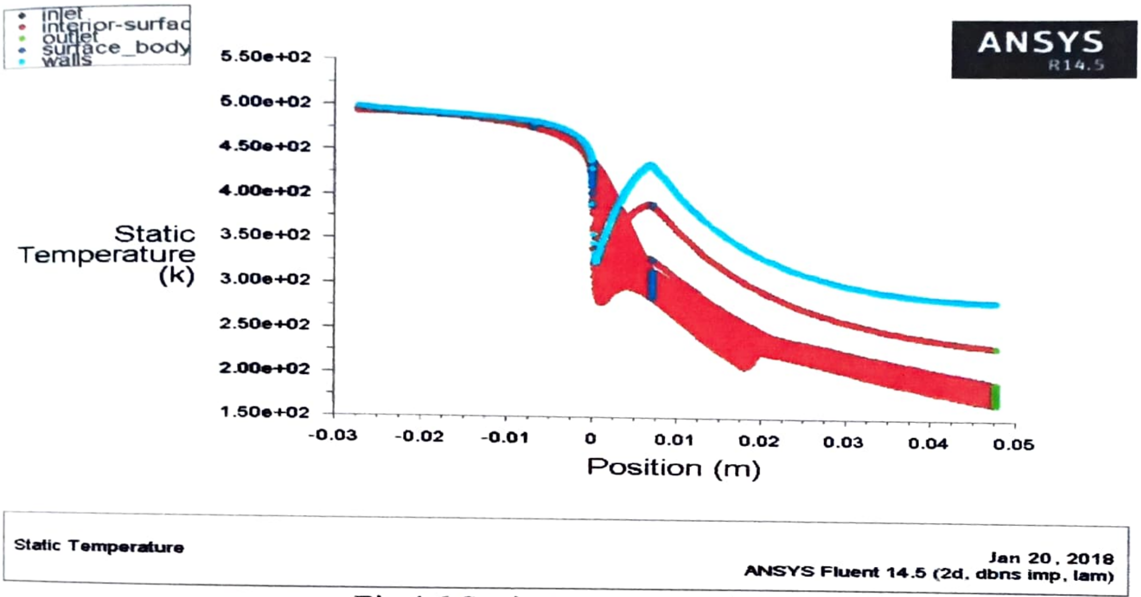


Fig 4.6 Static temperature plot

4.4.4 Contours of static pressure:

In combustion chamber max pressure according to De Laval nozzle principle and gradually decreases to convergent-divergent position and outlet. Due to backward pressure at diffusion chamber so sudden drop in pressure than turbulent shock or observed.

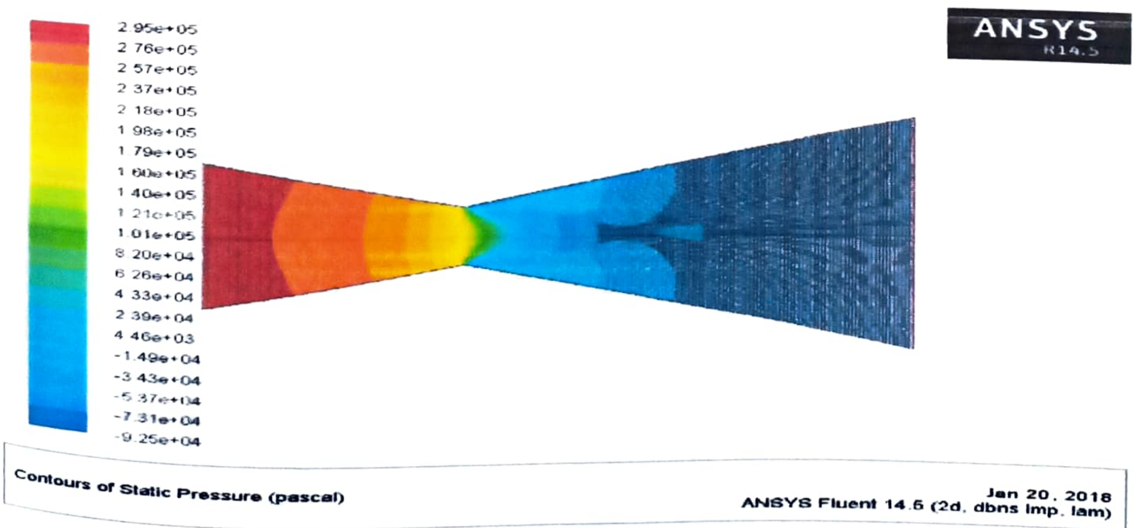


Fig 4.7 Contours of Pressure

4.4.5 Contours of static temperature:

Temperature boundary conditions were clearly affected in combustion chamber as well as convergent-divergent duct. Max temperature increase in inlet and duct due to compressed fluid and shocks at 3.30e+03.

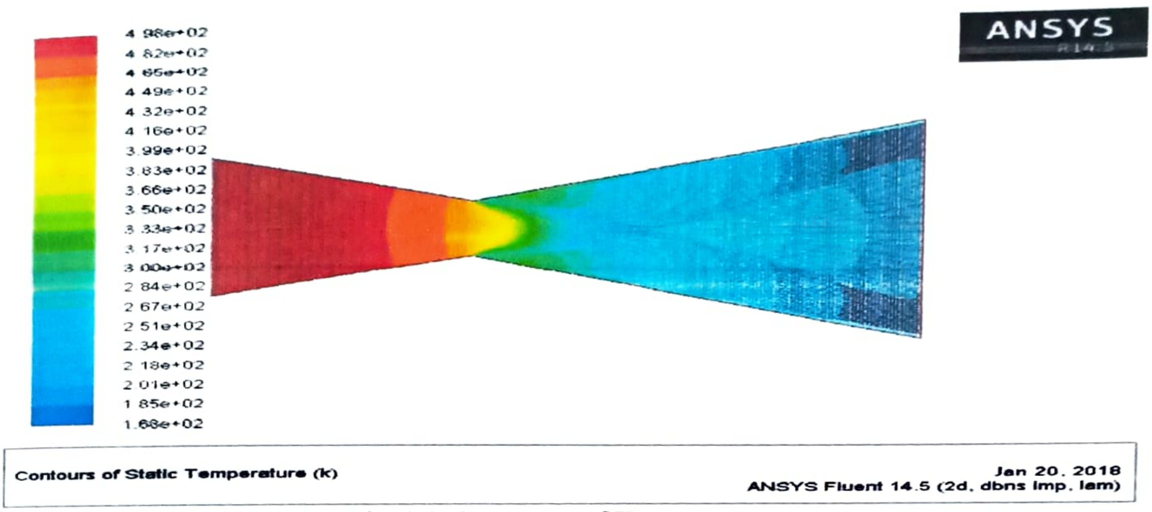


Fig 4.8 Contours of Temperature

4.4.6 Contours of velocity:

At given boundary conditions, nozzle Mach no lower value at inlet and higher value at outlet with range 1.06e-01 to 2.11e+00.

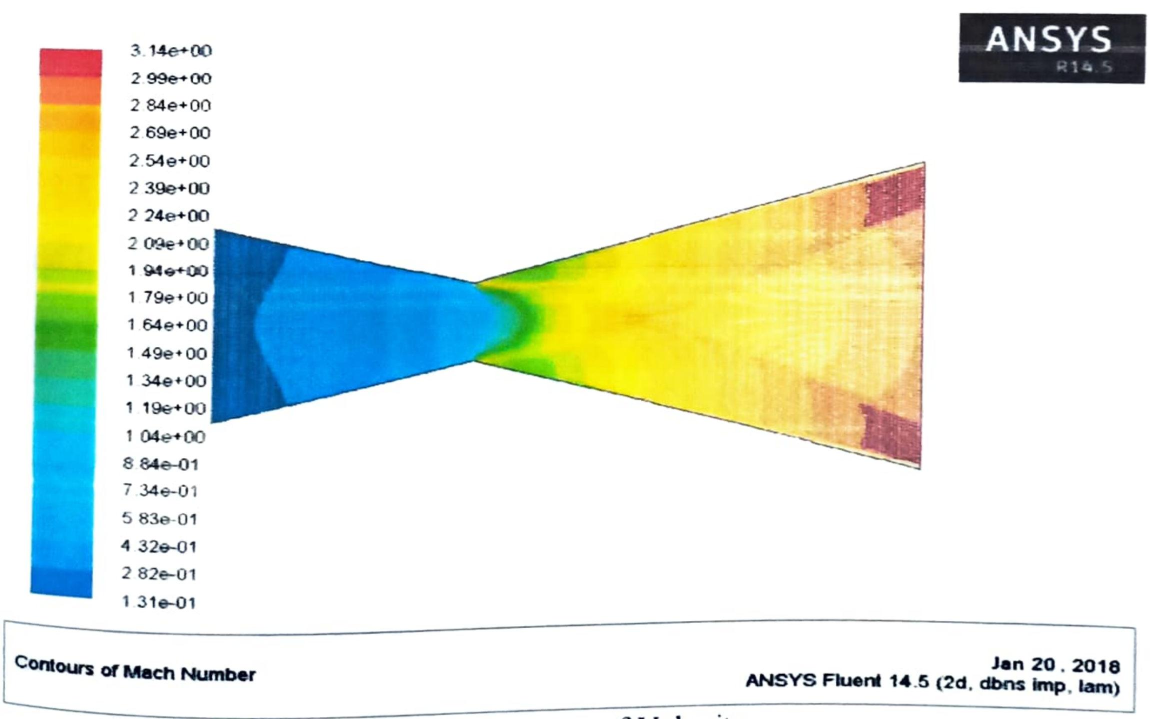


Fig 4.9 Contours of Velocity

5 ANALYSIS OF ROCKET NOZZLE

5.1 Introduction:

Various contours and plots have been checked to obtain the better divergence angle for a rocket nozzle. Plots such as velocity plot, pressure plot, temperature plot and also graphs are also plotted to obtain the best conditions.

5.2 CFD Analysis of 10^0 Rocket Nozzle:

The nozzle is designed as per the requirement with required dimensions, angles of inlet, outlet and walls.

5.2.1 Geometry:

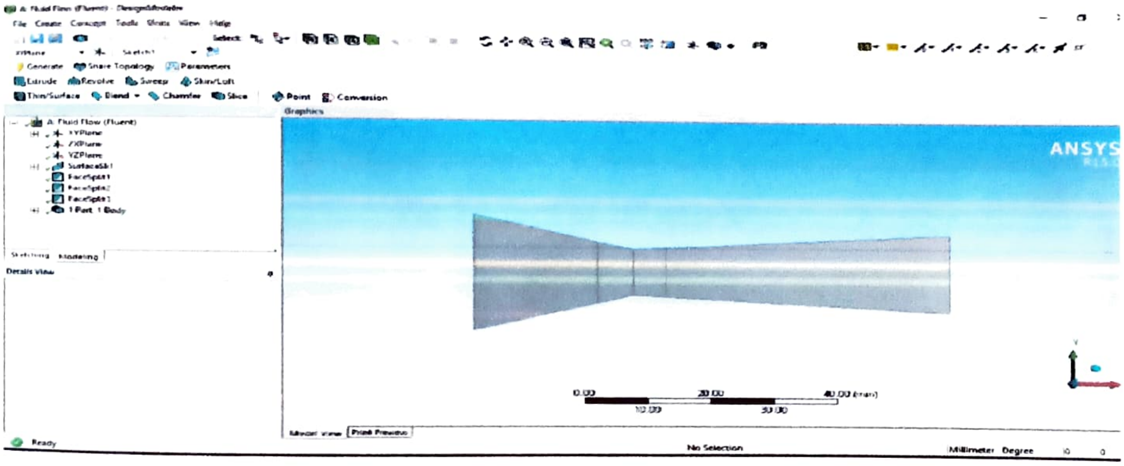


Fig 5.1 Geometry of 10^0 Rocket Nozzle

5.2.2 Meshing:

Next is obtained geometry model is meshed by using fluent software.

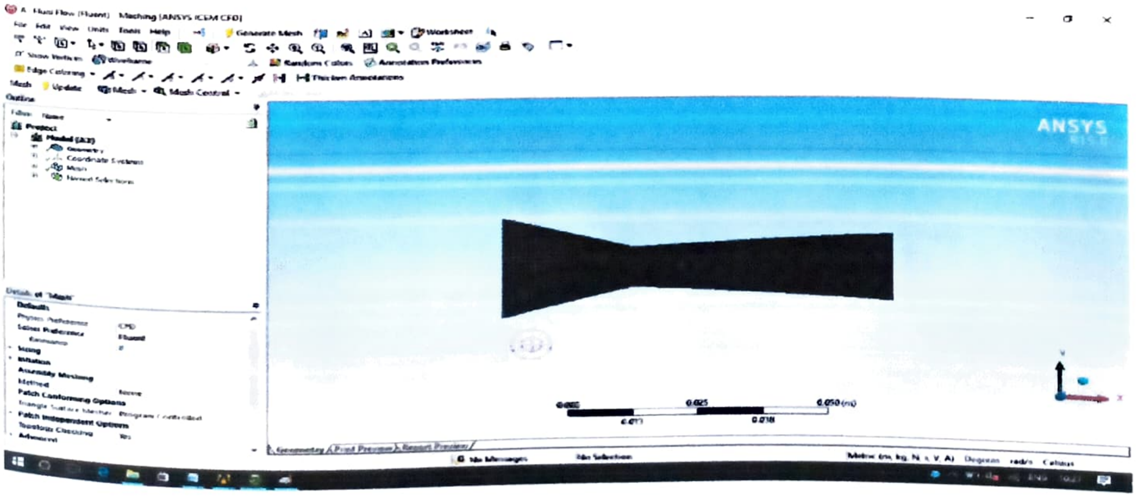


Fig 5.2 Meshed model of 10^0 nozzle

5.2.3 Specifying Boundaries for Inlet and Outlet:

Inlet:

The selections are named for further purpose i.e. to give boundary conditions, etc, etc…

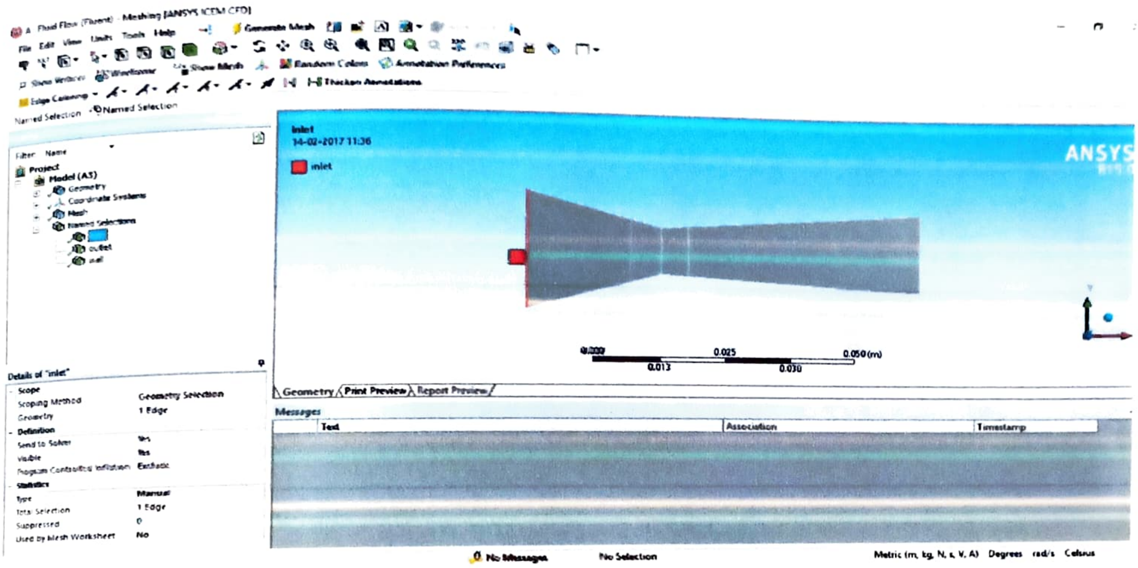


Fig 5.3 Inlet of 10^0 nozzle

Outlet:

Here the outlet is marked.

Fig 5.4 Outlet of 10^0 nozzle

Walls:

Named selection wall is specified.

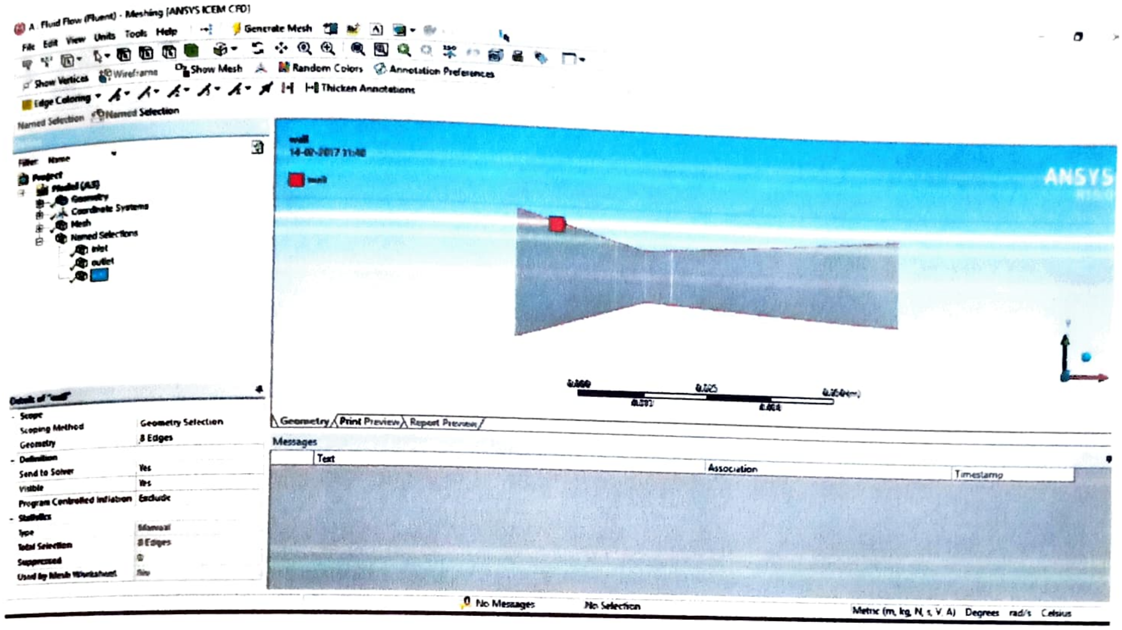
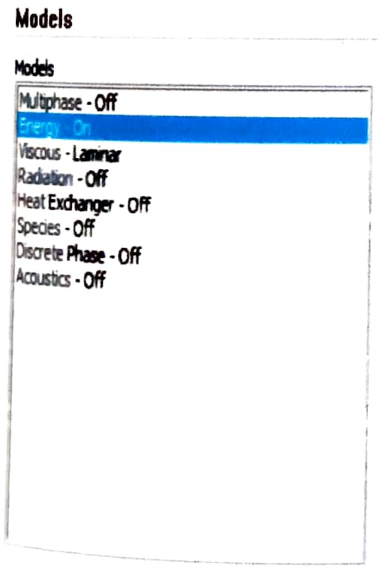
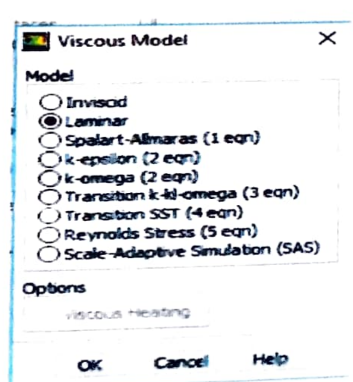


Fig 5.5 Walls of 10^0 Nozzle





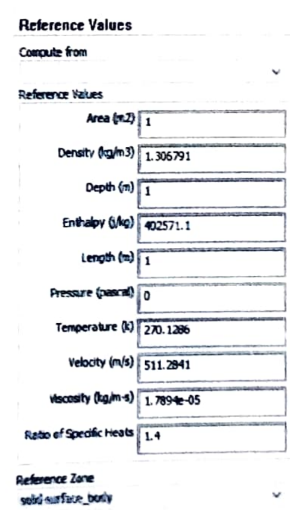


Fig 5.6 Energy Equation(ON) Fig 5.7 Viscous model Fig 5.8 Reference Values

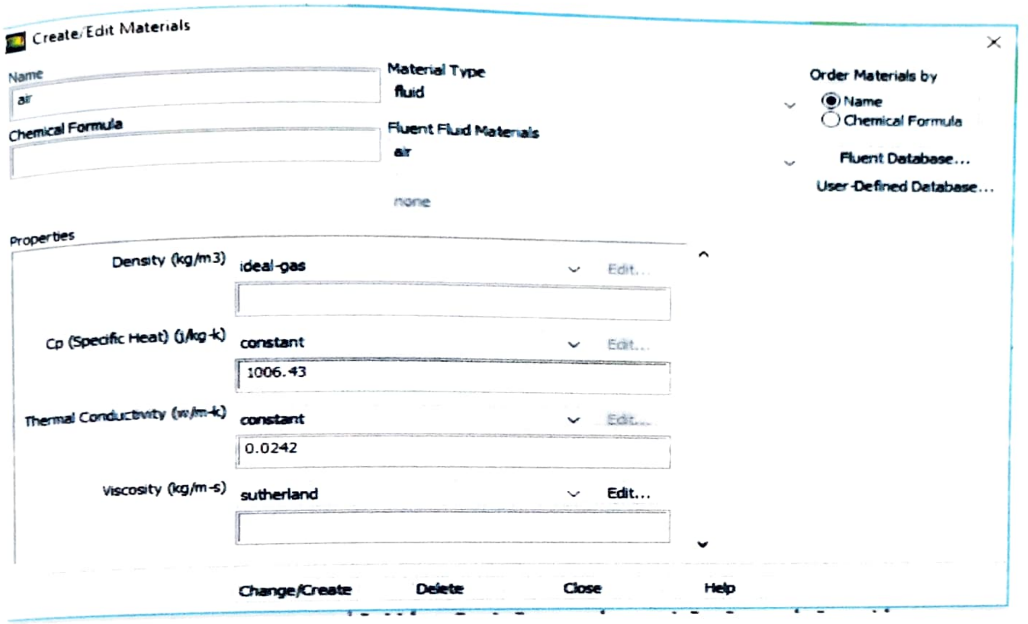


Fig 5.9 Material Properties

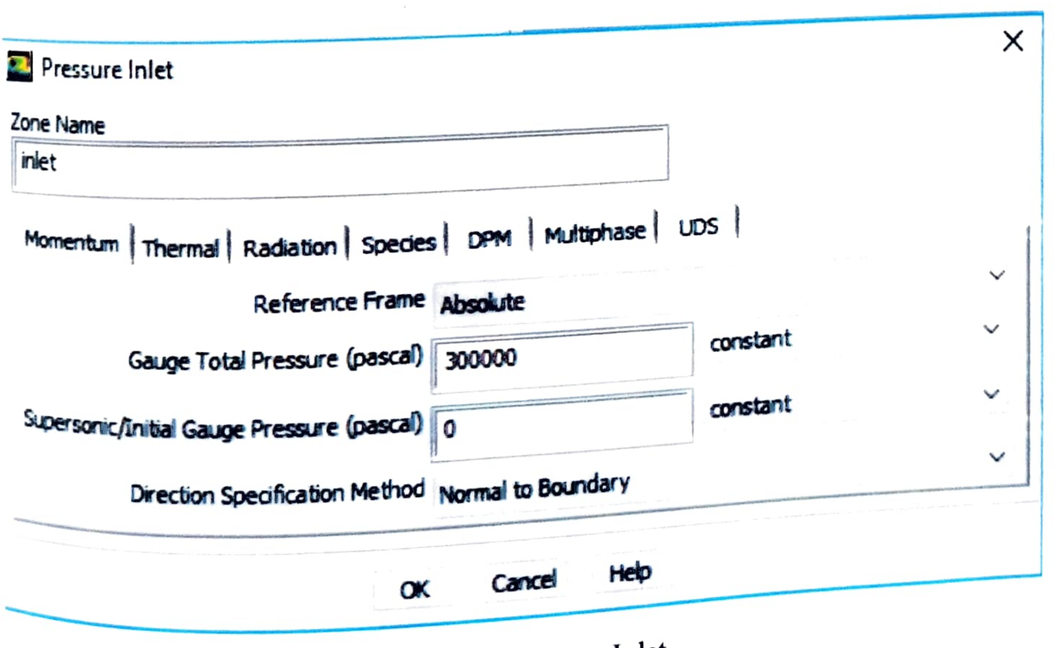


Fig 5.10 Pressure Inlet

The designed rocket nozzle is made to run and the solution is converged at 607 iterations.

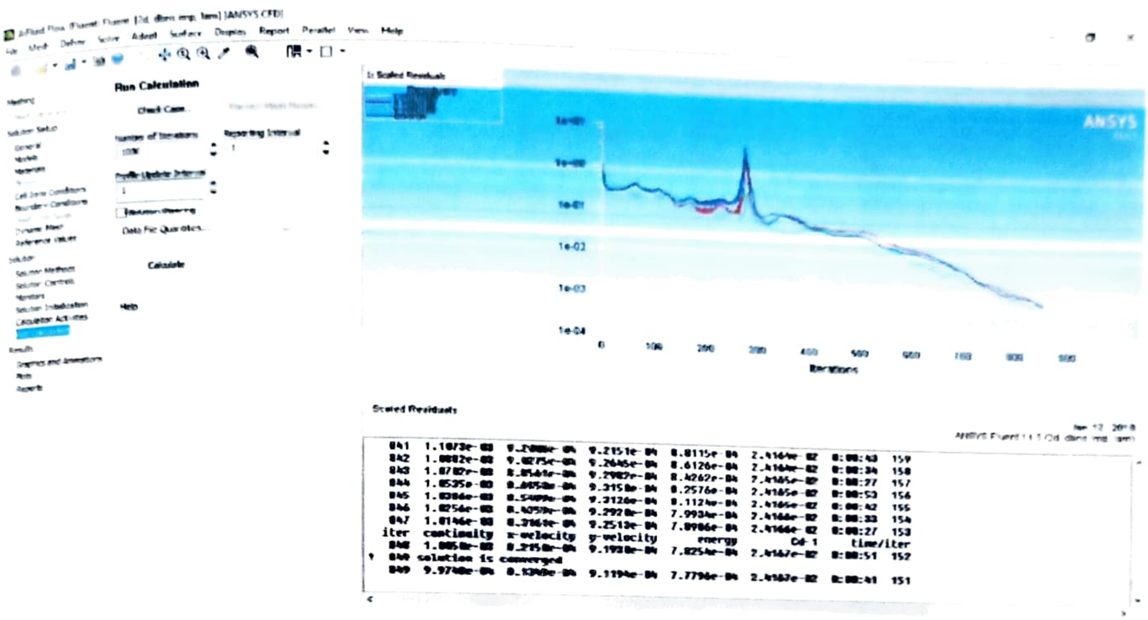
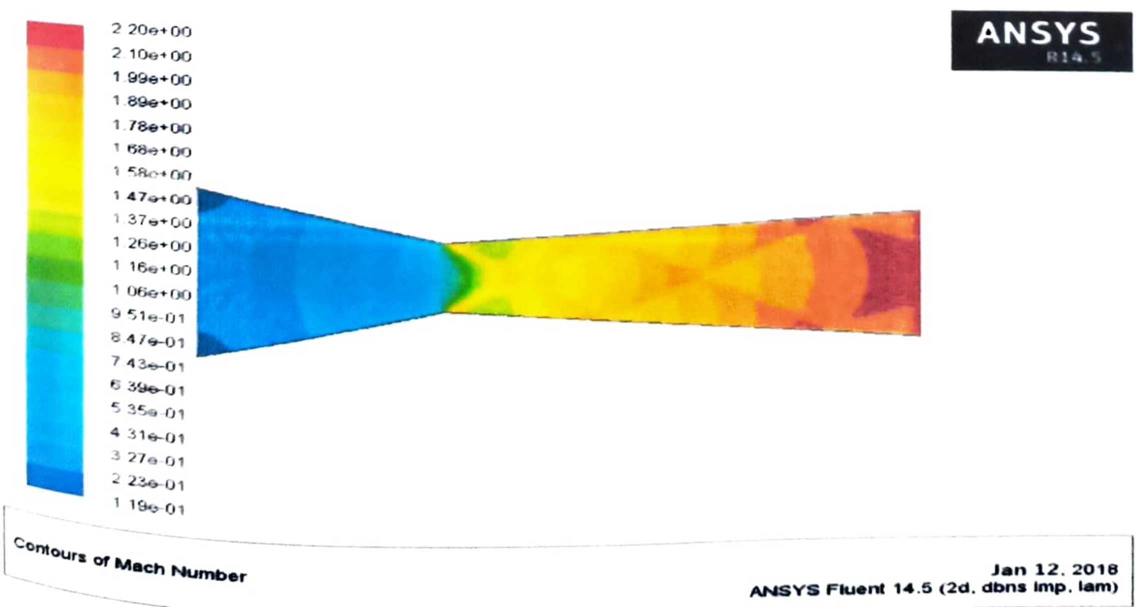


Fig 5.11 Iterations

5.2.4 Contour of Velocity:

It is clearly observed from the figure that the Mach number is 0.14 i.e. sub-sonic region in convergent section at inlet point, at the throat the Mach number is 0.73 i.e. in sonic, at the exit it becomes supersonic for the designed nozzle. Near the wall, the Mach number is 2.07. This is due to the viscosity and the turbulence in the fluid



.Fig 5.12 Velocity contour for Nozzle with angle 10^0

5.2.5 Pressure Contour:

Static pressure is the pressure that is exerted by a fluid. Specifically, it is the pressure measured when the fluid is still, or at rest. The below figure reveals the fact that gas gets expanded in the nozzle exit. The static pressure in the inlet is observed to be 3bar and as we move to the throat there is a decrease in the value. After the throat, the pressure falls in a more repaid manner towards the exit of the nozzle. At the exit it is found to be closure atmospheric.

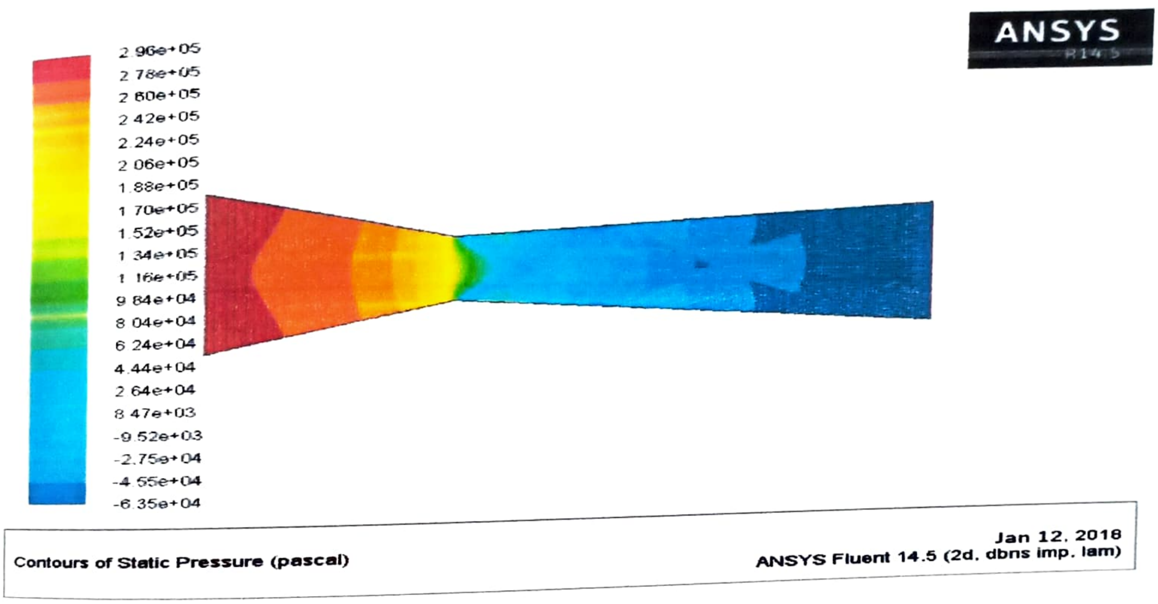


Fig 5.13 Pressure Contour for nozzle with divergence angle of 10^0

5.2.6 Temperature Contour:

The total temperature always remains a constant in the inlet up to the throat after which it tends to increase. Near the walls the temperature increases to 400K. The temperature at the throat is 319K and the temperature at the exit is 398K.

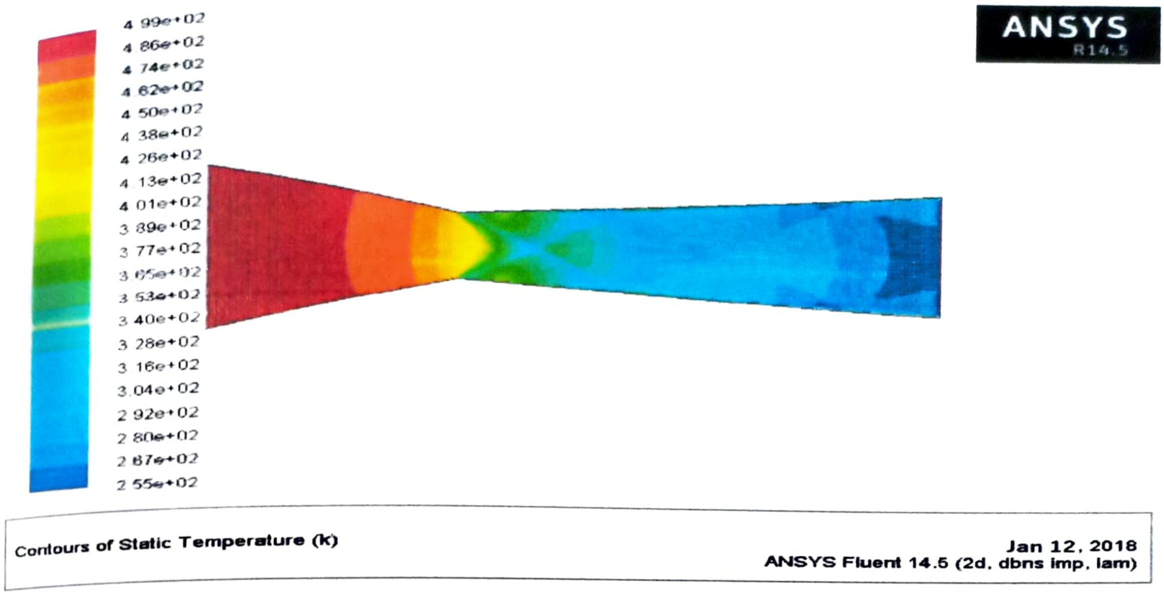


Fig 5.14 Temperature Contour for Nozzle with divergence angle 10^0

5.2.7 Velocity Plot:

A graph is plotted by taking position (m) on X-axis and Mach number on Y-axis. It is clearly observed that the velocity is increased.

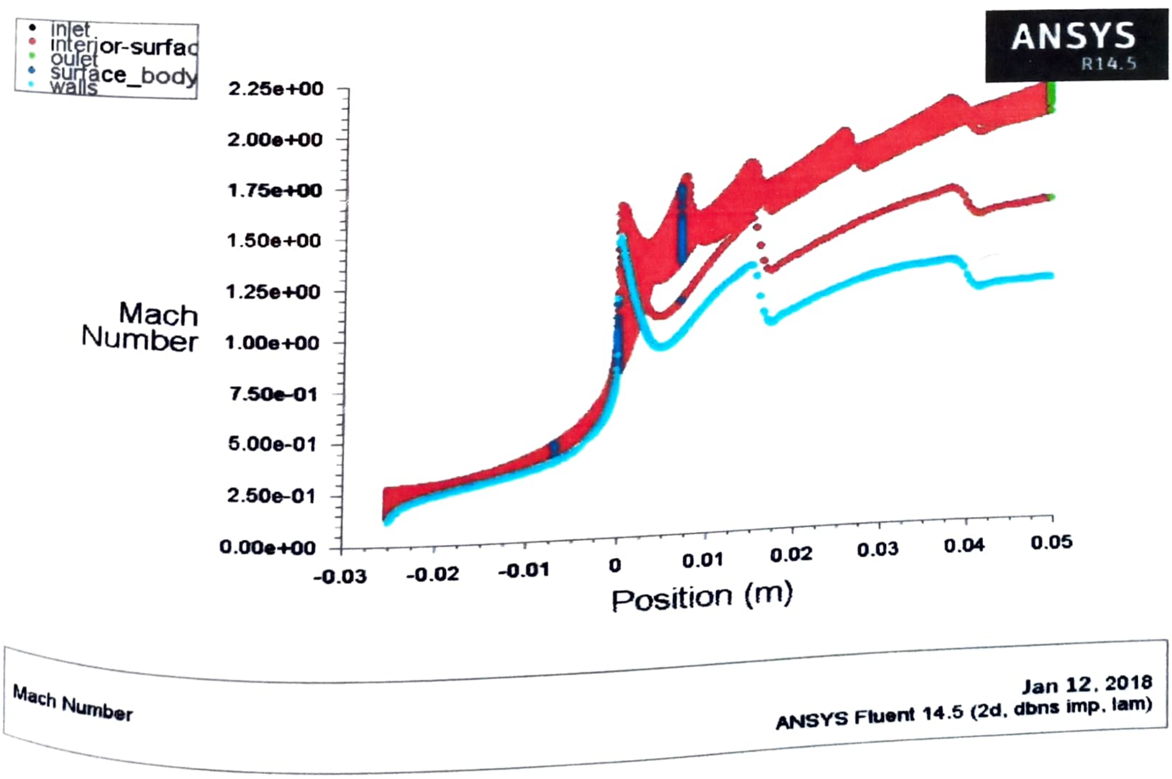


Fig 5.15 XY plot for velocity contour with divergence angle of 10^0

5.2.8 Pressure Plot:

A graph is plotted by taking position (m) on X-axis and Static Pressure (Pa) on Y-axis. There is decrease in the pressure from inlet to outlet.

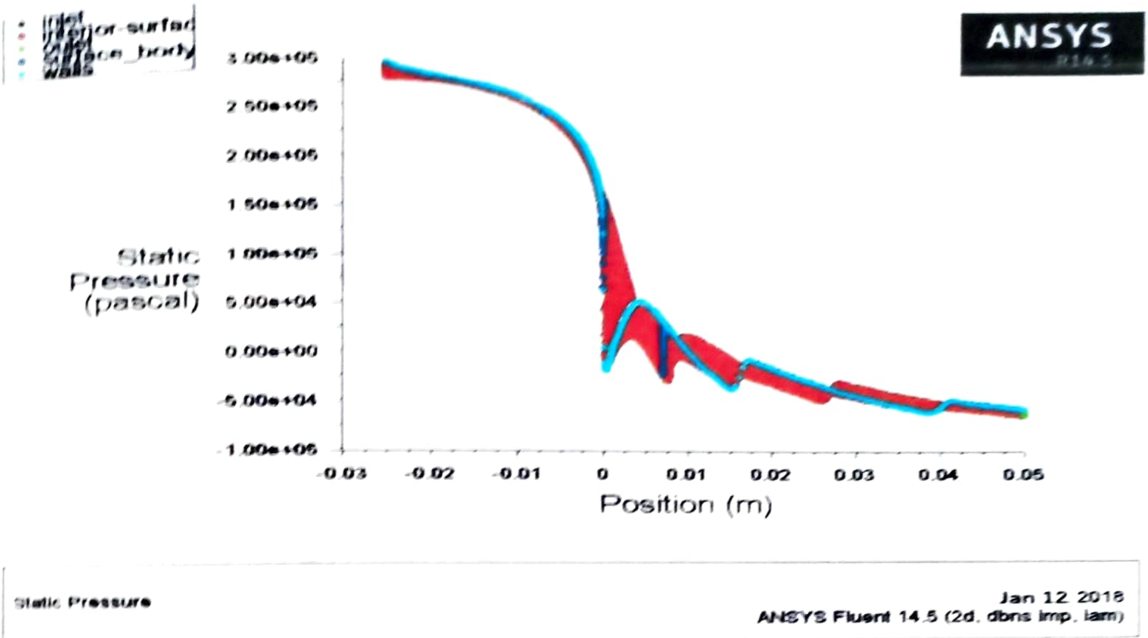


Fig 5.16 XY plot for pressure contour with divergence angle of 10^0

5.2.9 Temperature Plot:

A graph is plotted by taking position (m) on X-axis and Static temperature (k) on Y-axis. There is sudden increase and then decrease in temperature.

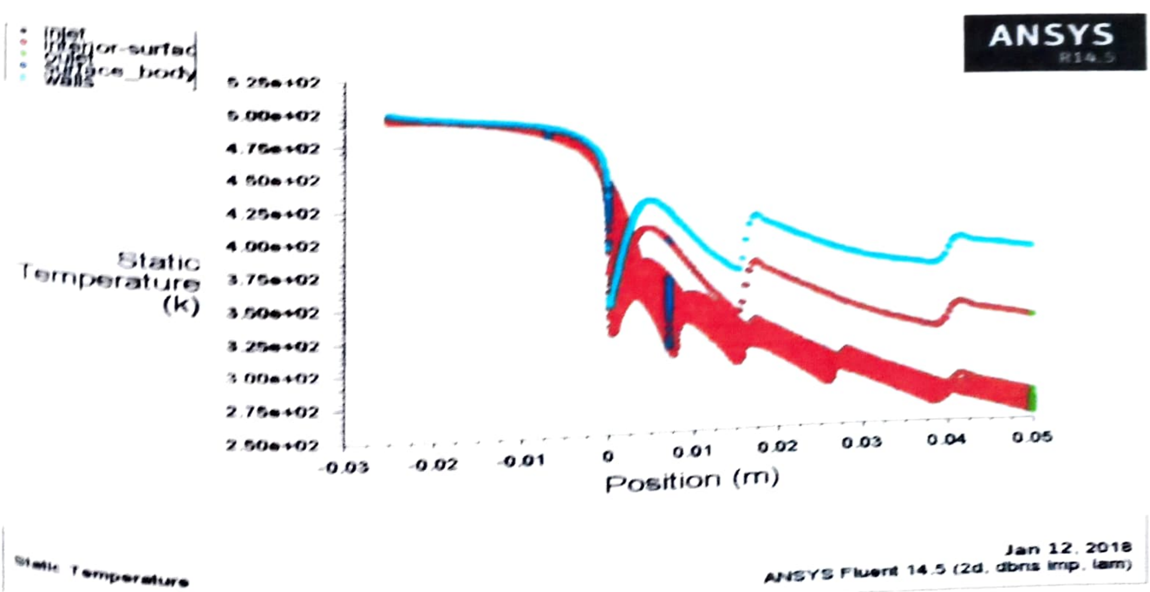


Fig 5.17 XY plot for static temperature with divergence angle of 10^0

5.3 CFD Analysis of 15⁰ Rocket Nozzle:

The nozzle is designed as per the requirement with required dimensions, angles of inlet, outlet and walls.

5.3.1 Geometry:

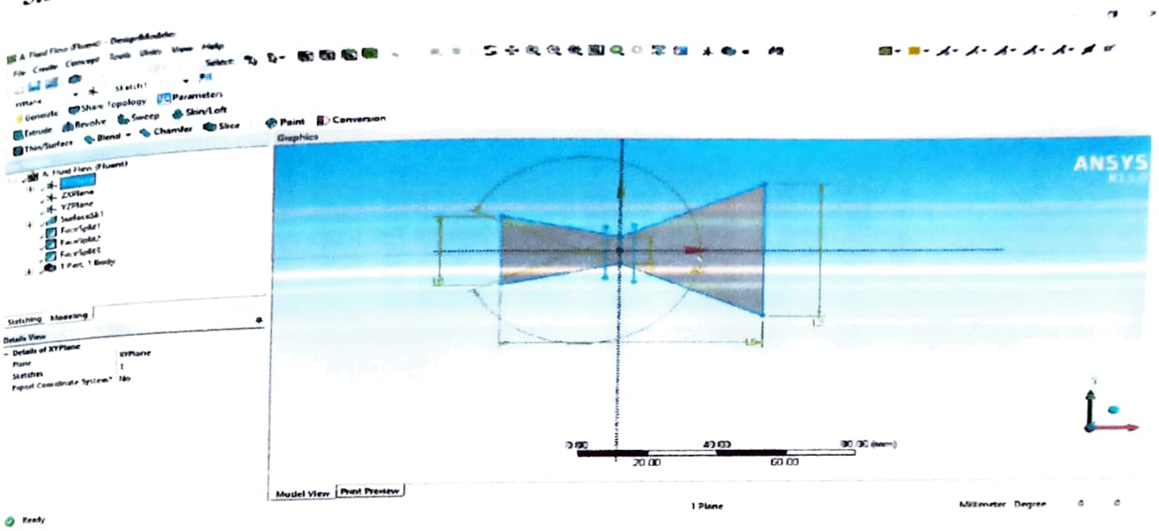


Fig 5.45 Geometry of 15^0 Rocket Nozzle

5.3.2 Meshing:

Next is obtained geometry model is meshed by using fluent software.

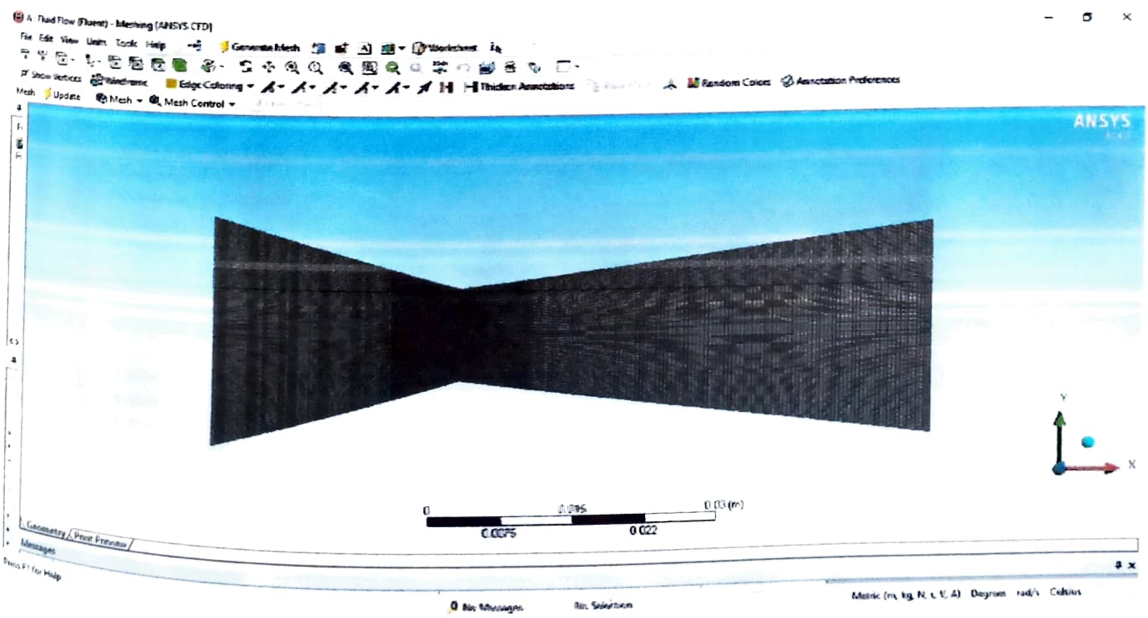


Fig 5.46 Meshed model of 15^0 nozzle

The designed rocket nozzle is made to run and solution is converged at 606 iterations.

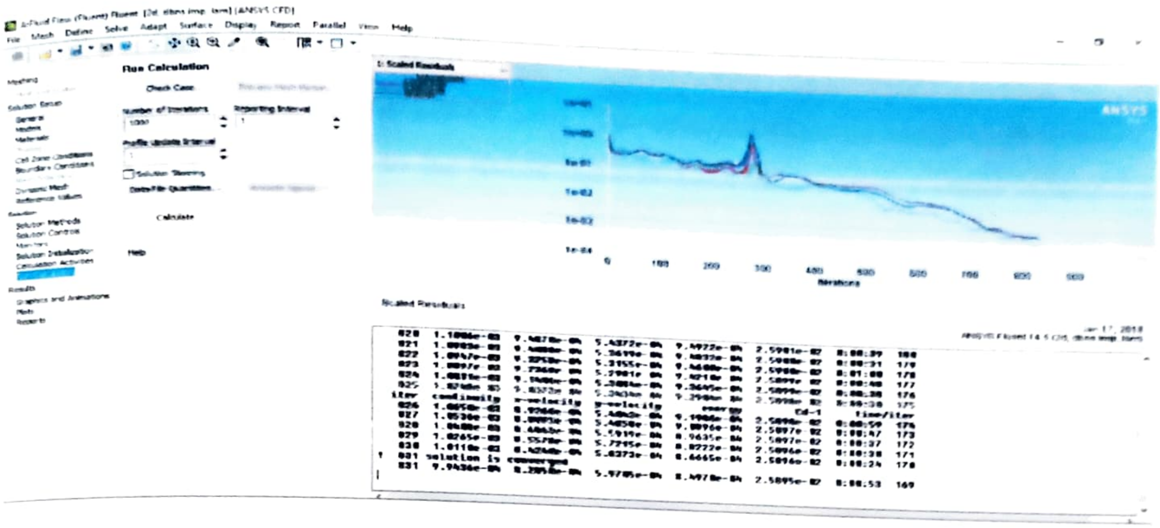


Fig 5.47 Iterations

5.3.3 Contour of Velocity:

It is clearly observed from the figure that the Mach number is 0.049 i.e. sub-sonic region in convergent section at inlet point, at the throat the Mach number is 0.97 i.e. in sonic, at the exit it becomes supersonic for the designed nozzle. Near the wall, the Mach number is 3.15. This is due to the viscosity and the turbulence in the fluid.

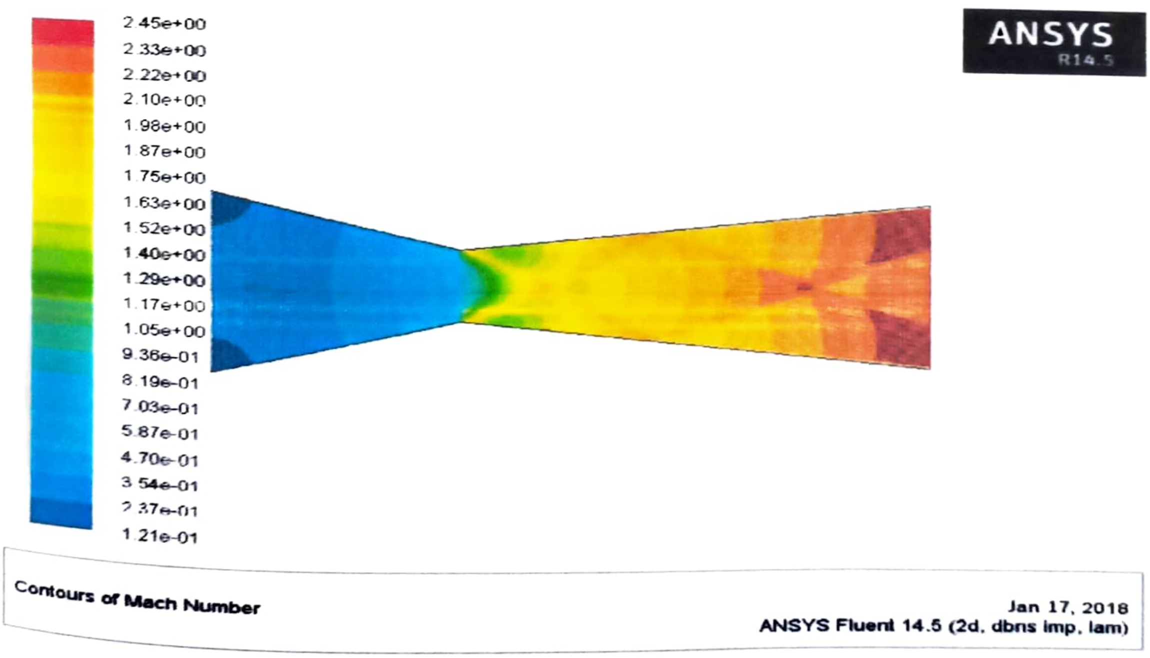


Fig 5.48 Velocity contour for Nozzle with angle 15^0

5.3.4 Pressure Contour:

Static pressure is the pressure that is exerted by a fluid. Specifically, it is the pressure measured when the fluid is still, or at rest. The above figure reveals the fact that gas gets expanded in the nozzle exit. The static pressure in the inlet is observed to be 3bar and as we move to the throat there is a decrease and the value is found to be 1.60bar. After the throat, the pressure falls in a more repaid manner towards the exit of the nozzle. At the exit it is found to be 2.94bar

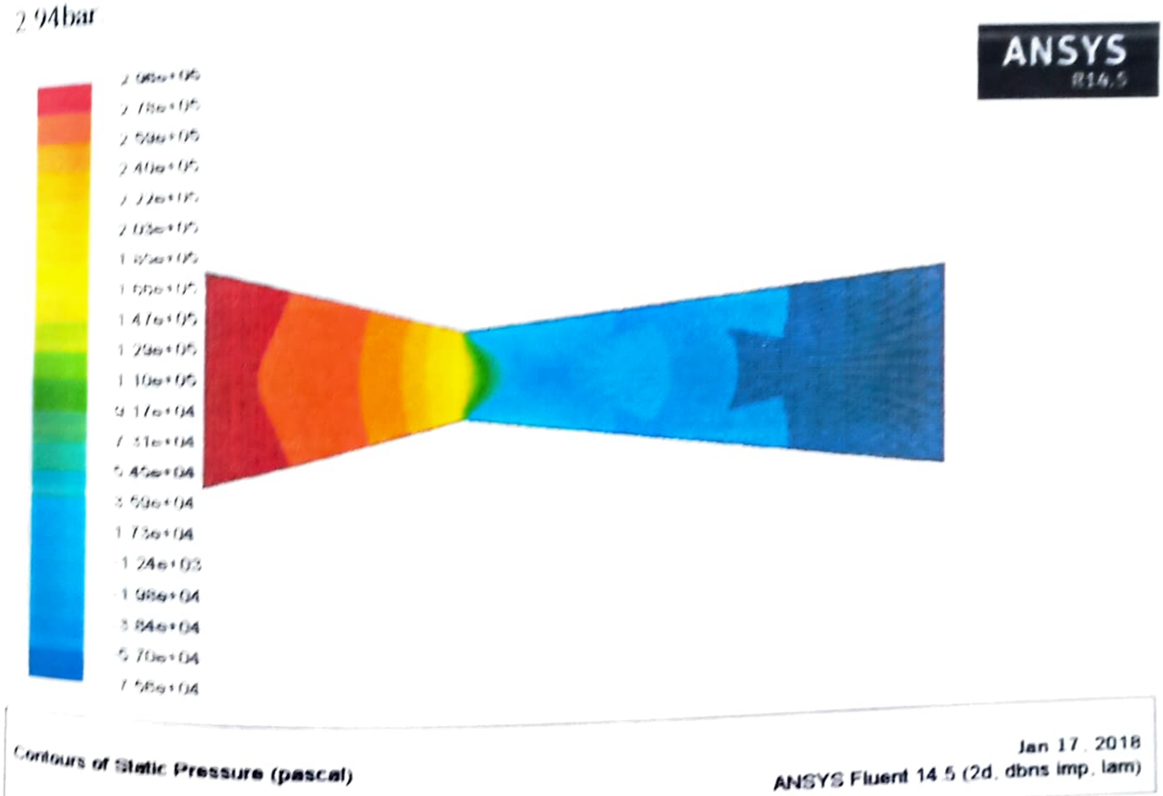


Fig 5.49 Pressure Contour for nozzle with divergence angle of 15^0

5.3.5 Temperature Contour:

The total temperature always remains a constant in the inlet up to the throat after which it tends to increase. Near the walls the temperature increases to 400 K. In the inlet and the throat the temperature is 426 K. After the throat the temperature increases to 317K at the exit.

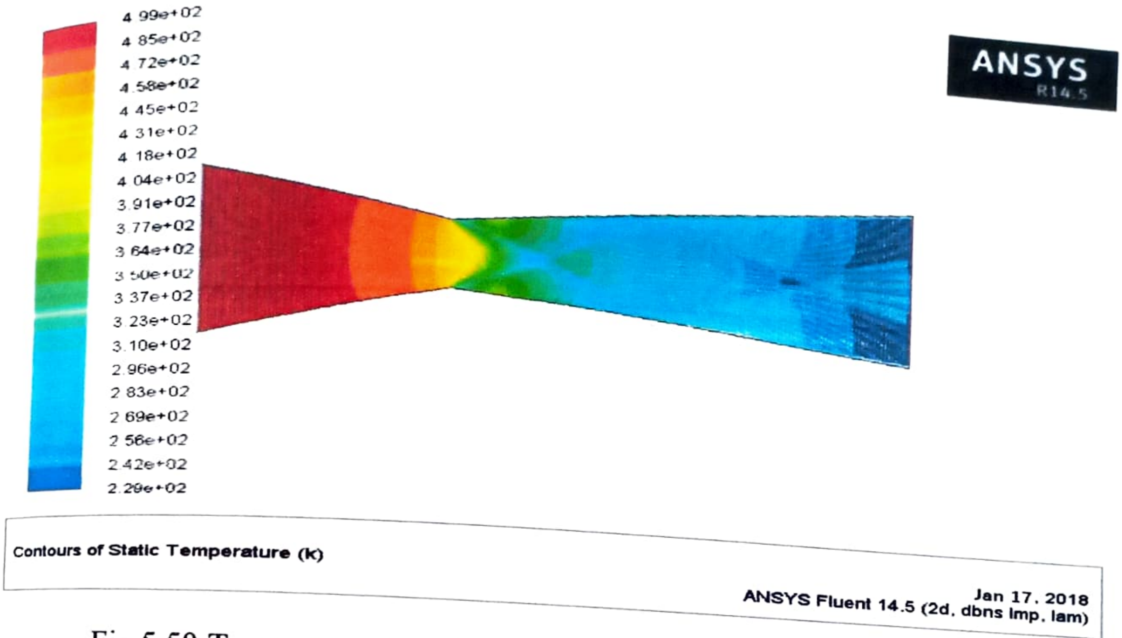


Fig 5.50 Temperature Contour for Nozzle with divergence angle 15^0

5.3.6 Velocity Plot:

A graph is plotted by taking position (m) on X-axis and Mach number on Y-axis. It is clearly observed that the velocity is increased.

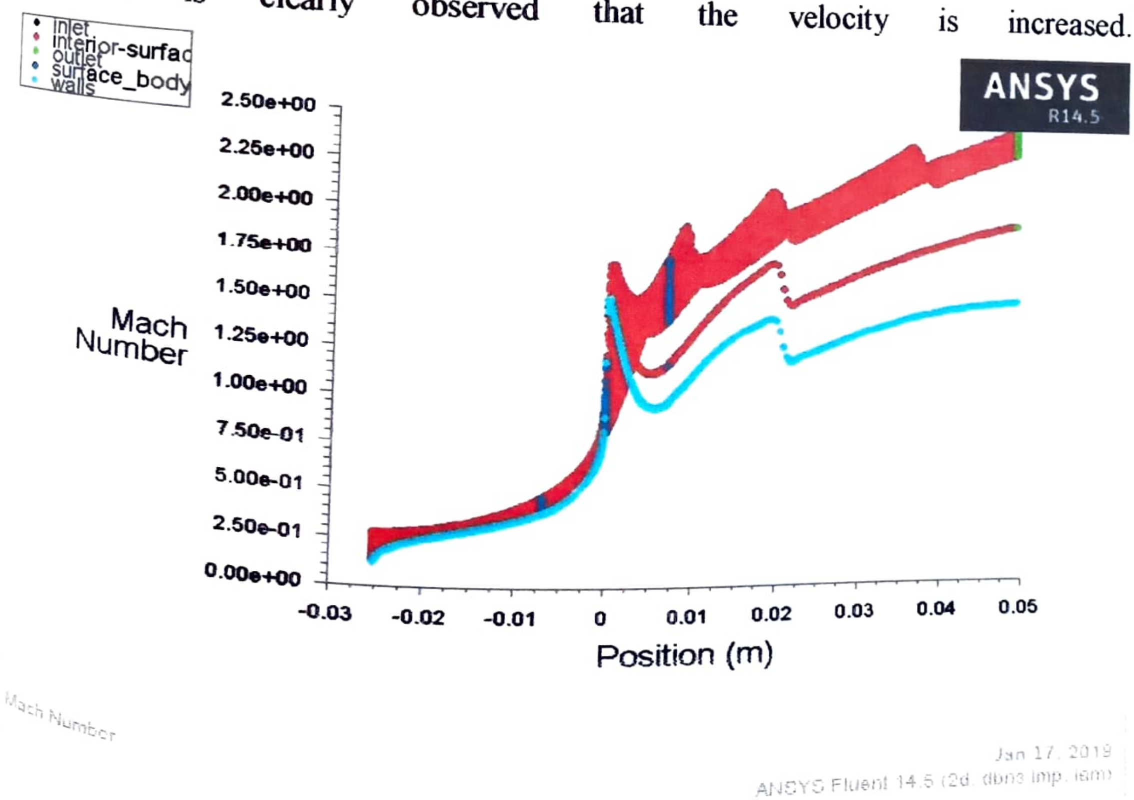


Fig 5.51 XY plot for velocity contour with divergence angle of 15^0

5.3.7 Pressure Plot:

A graph is plotted by taking position (m) on X-axis and Static Pressure (Pa) on Y-axis. There is decrease in the pressure from inlet to outlet.

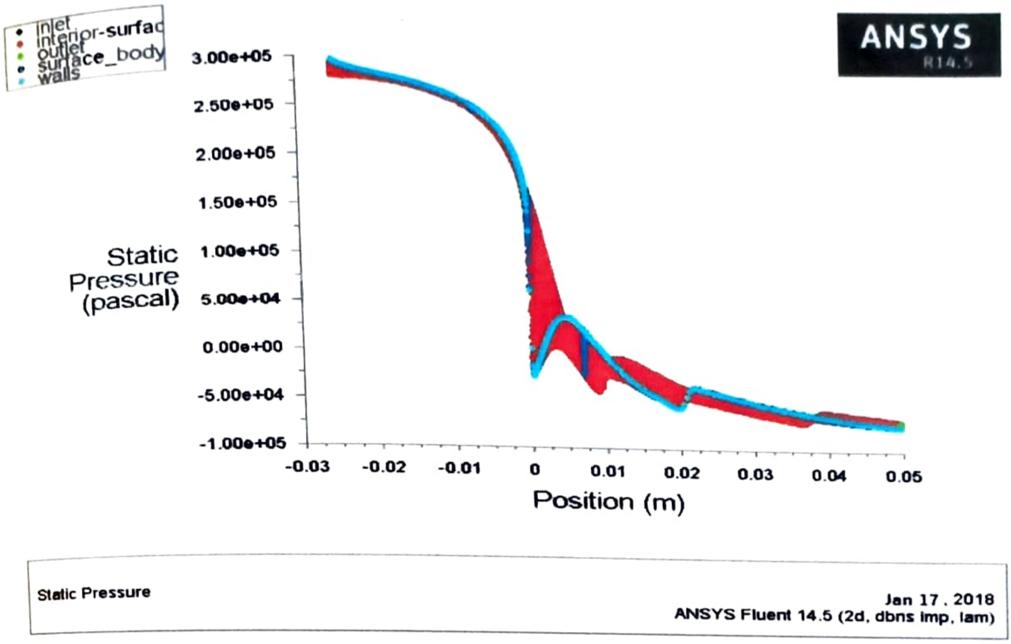


Fig 5.52 XY plot for pressure contour with divergence angle of 15^0

5.3.8 Temperature Plot:

A graph is plotted by taking position (m) on X-axis and Static temperature (k) on Y-axis. There is sudden increase and then decrease in temperature.

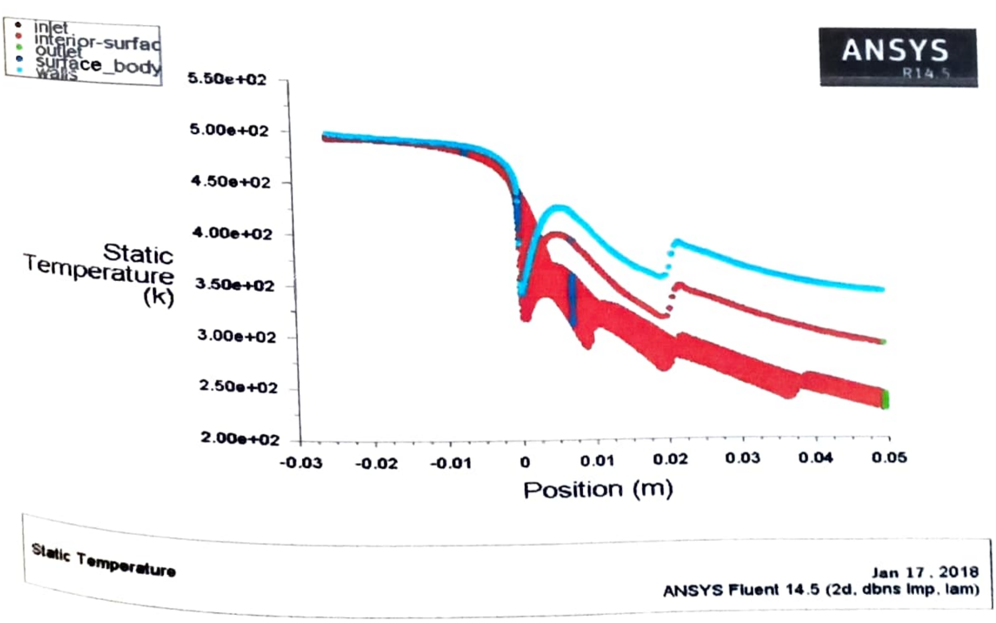


Fig 5.53 XY plot for static temperature with divergence angle of 15^0

5.4 CFD Analysis of 20^0 Rocket Nozzle:

The nozzle is designed as per the requirement with required dimensions, angles of inlet, outlet and walls.

5.4.1 Geometry:

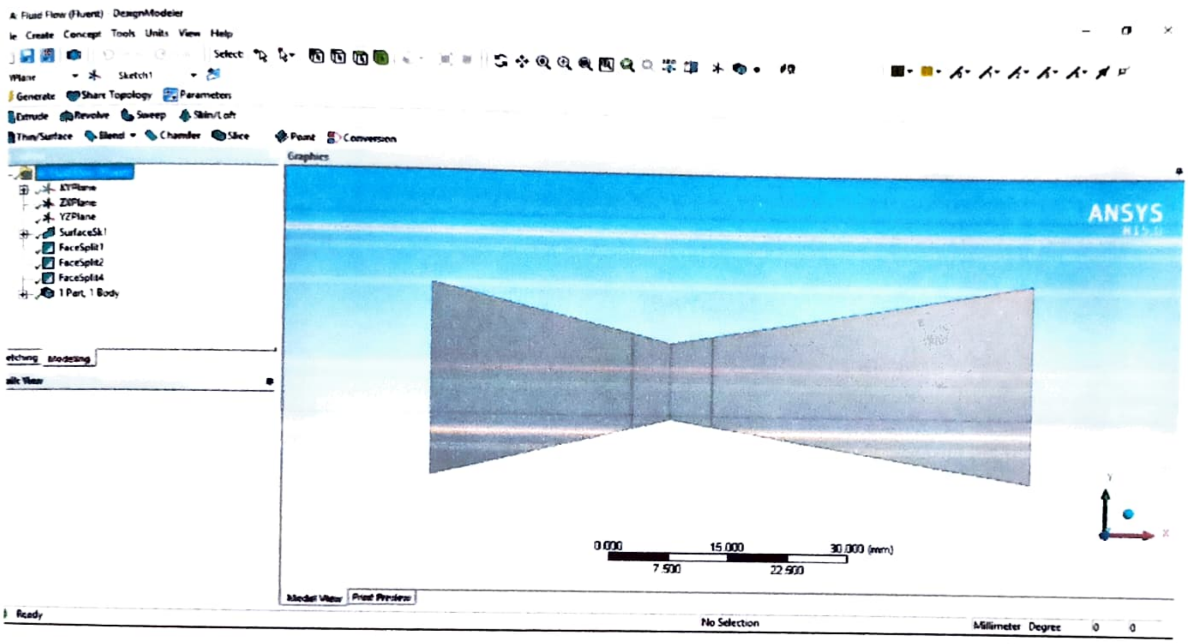


Fig 5.18 Geometry of 20^0 Rocket Nozzle

5.4.2 Meshing:

Next is obtained geometry model is meshed by using fluent software.

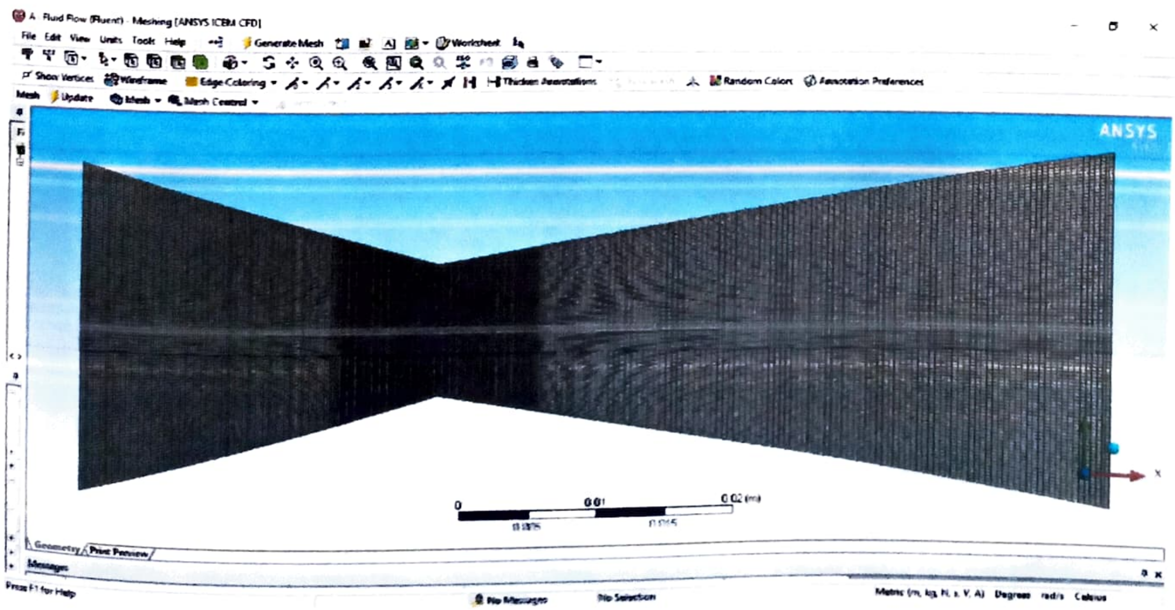


Fig 5.19 Meshed model of 20^0 nozzle

The designed rocket nozzle is made to run and calculate and the solution is converged at 723 iterations.

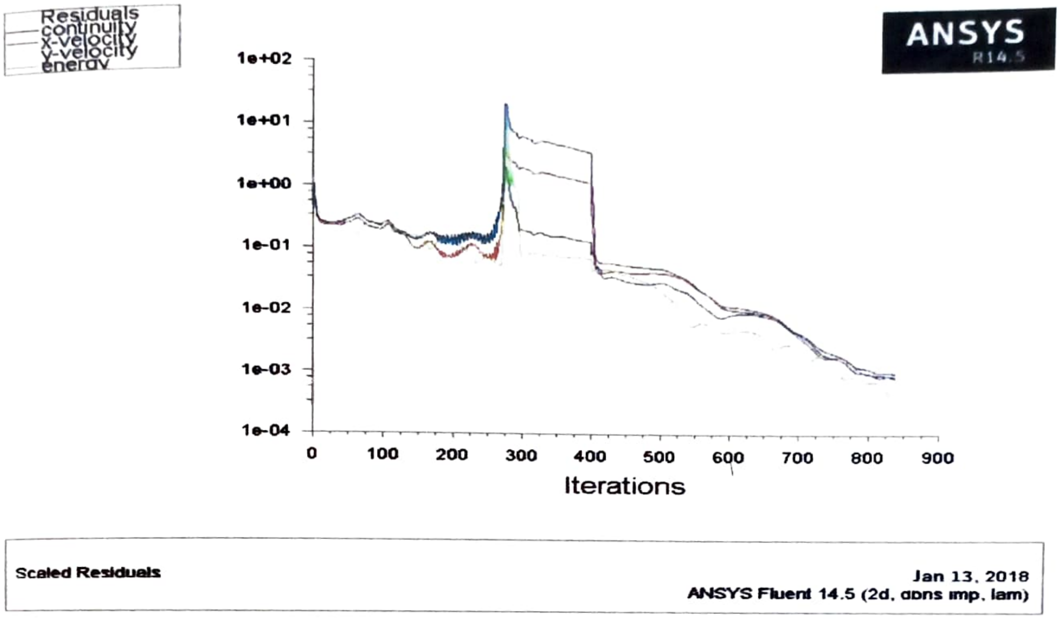


Fig 5.20 Iterations

5.4.3 Contour of Velocity:

It is clearly observed from the figure that the Mach number is 0.13 i.e. sub-sonic region in convergent section at inlet point, at the throat the Mach number is 2.68 i.e. in sonic, at the exit it becomes supersonic for the designed nozzle. Near the wall, the Mach number is 2.68. This is due to the viscosity and the turbulence in the fluid.

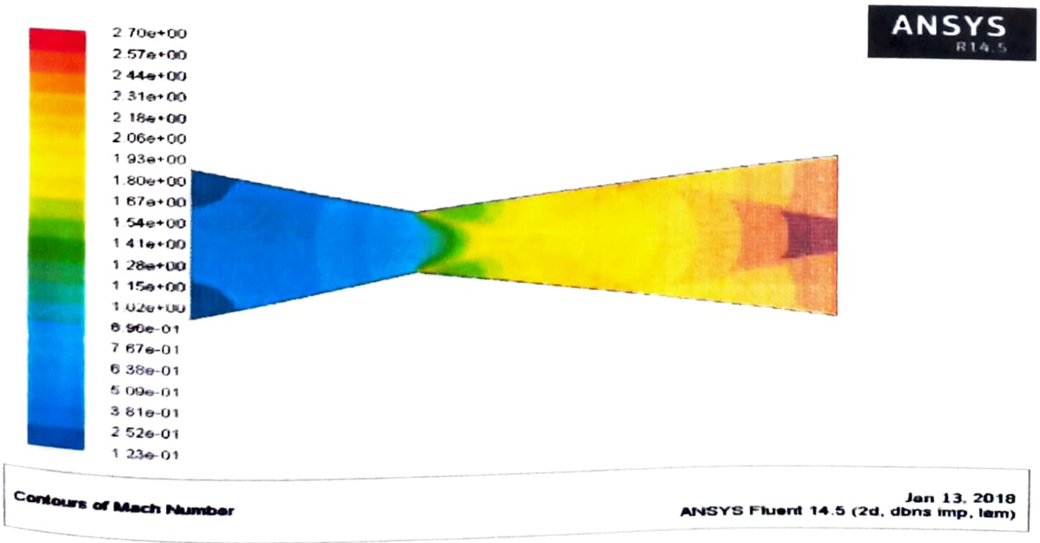


Fig 5.21 Velocity contour for Nozzle with angle 20^0

5.4.4 Pressure Contour:

Static pressure is the pressure that is exerted by a fluid. Specifically, it is the pressure measured when the fluid is still, or at rest. The above figure reveals the fact that gas gets expanded in the nozzle exit. The static pressure in the inlet is observed to be 3bar and as we move to the throat there is a decrease in the value. After the throat, the pressure falls in a more repaid manner towards the exit of the nozzle. At the exit it is found to be 2.87bar.

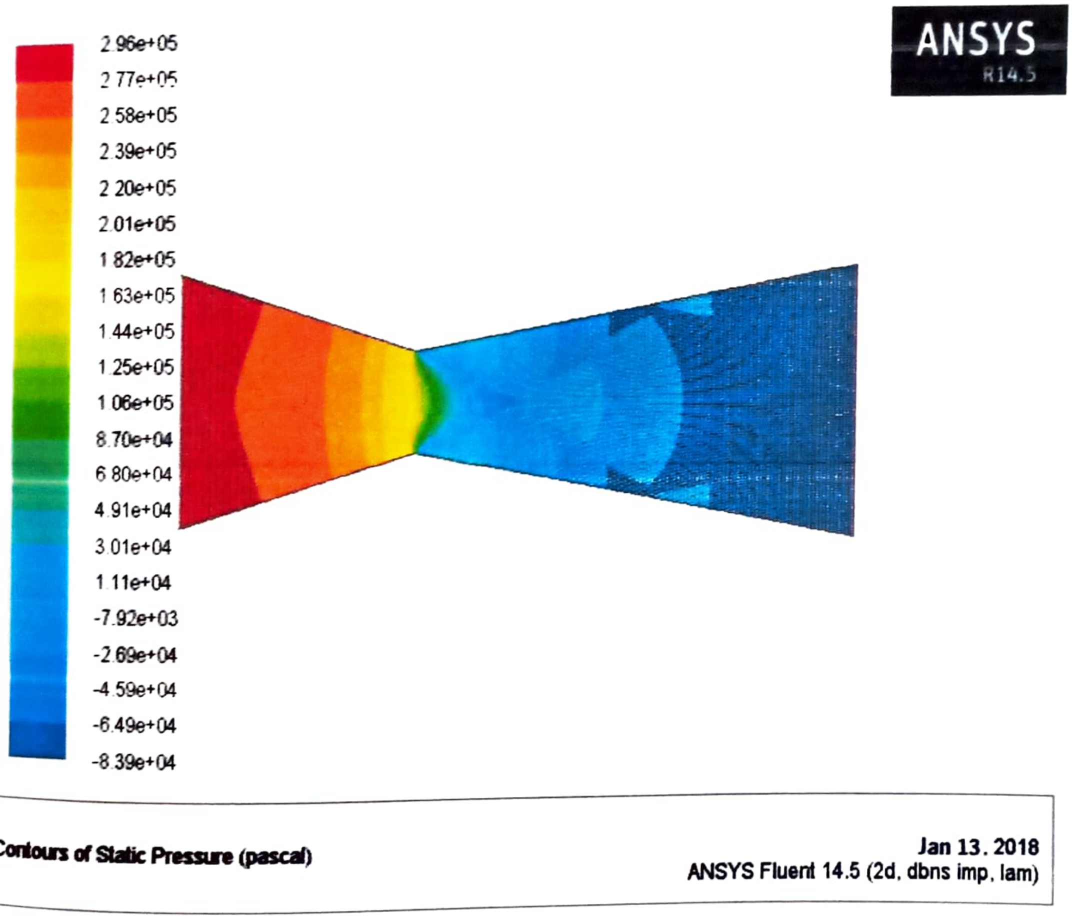


Fig 5.22 Pressure Contour for nozzle with divergence angle of 20^0

5.4.5 Temperature Contour:

The total temperature always remains a constant in the inlet up to the throat after which it tends to increase. Near the walls the temperature increases to 400K. In the inlet and the throat the temperature is 305K. After the throat the temperature increases to 398.6K at the exit.

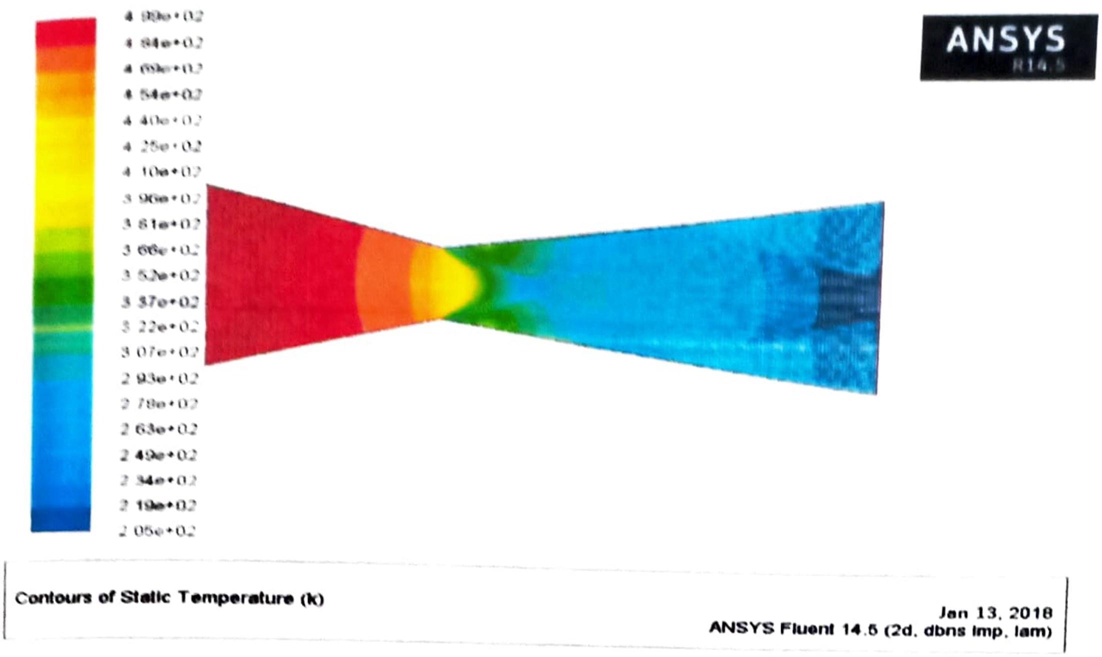


Fig 5.23 Temperature Contour for Nozzle with divergence angle 20^0

5.4.6Velocity Plot:

A graph is plotted by taking position (m) on X-axis and Mach number on Y-axis. It is clearly observed that the velocity is increased.

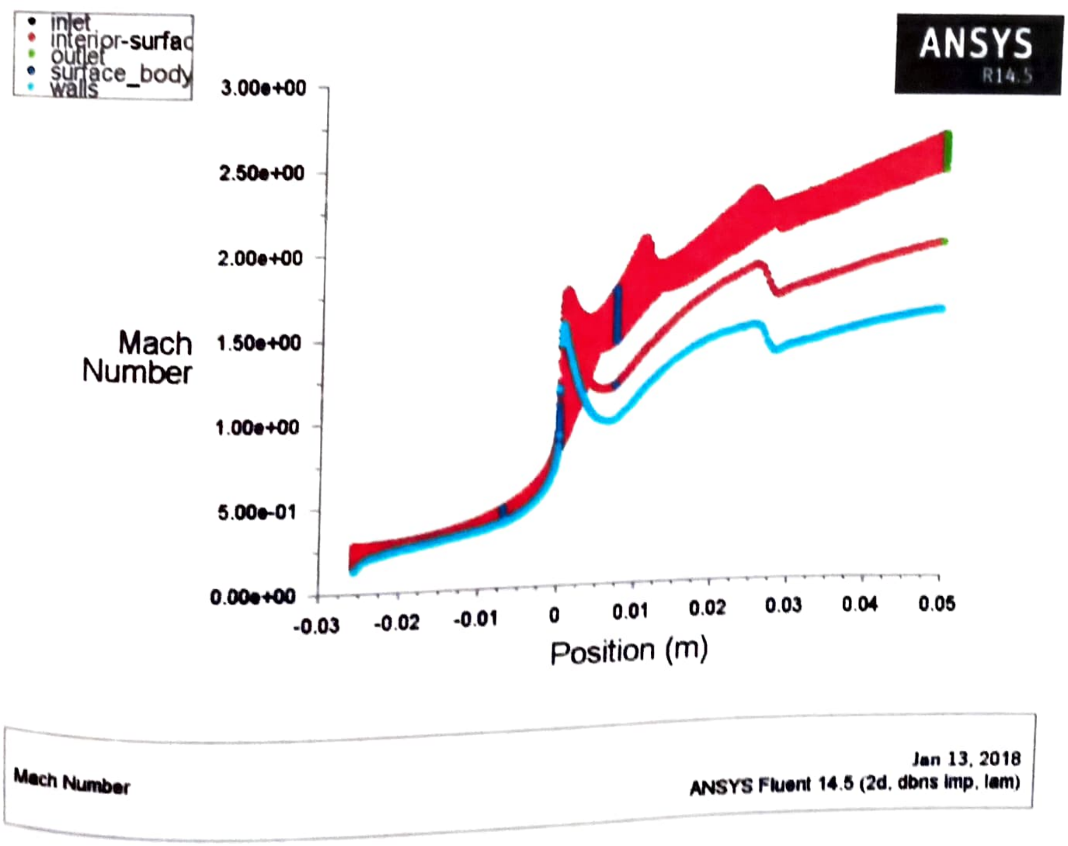


Fig 5.24 XY plot for velocity contour with divergence angle of 20^0

5.4.7 Pressure Plot:

A graph is plotted by taking position (m) on X-axis and Static Pressure (Pa) on Y-axis. There is decrease in the pressure from inlet to outlet.

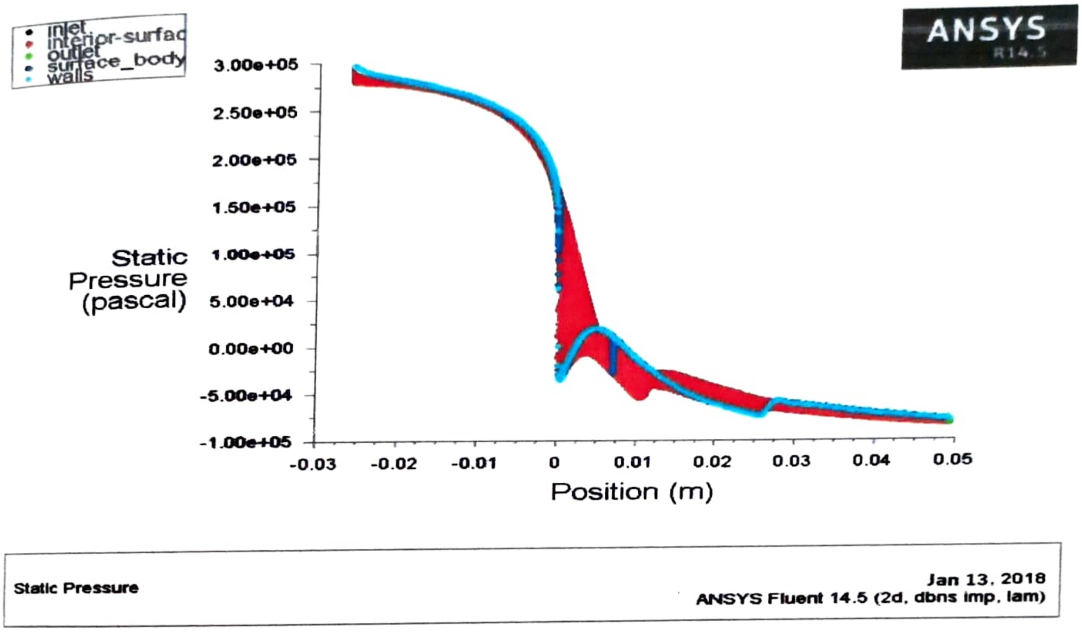


Fig 5.25 XY plot for pressure contour with divergence angle of 20^0

5.4.8 Temperature Plot:

A graph is plotted by taking position (m) on X-axis and Static temperature (k) on Y-axis.

There is sudden increase and then decrease in temperature.

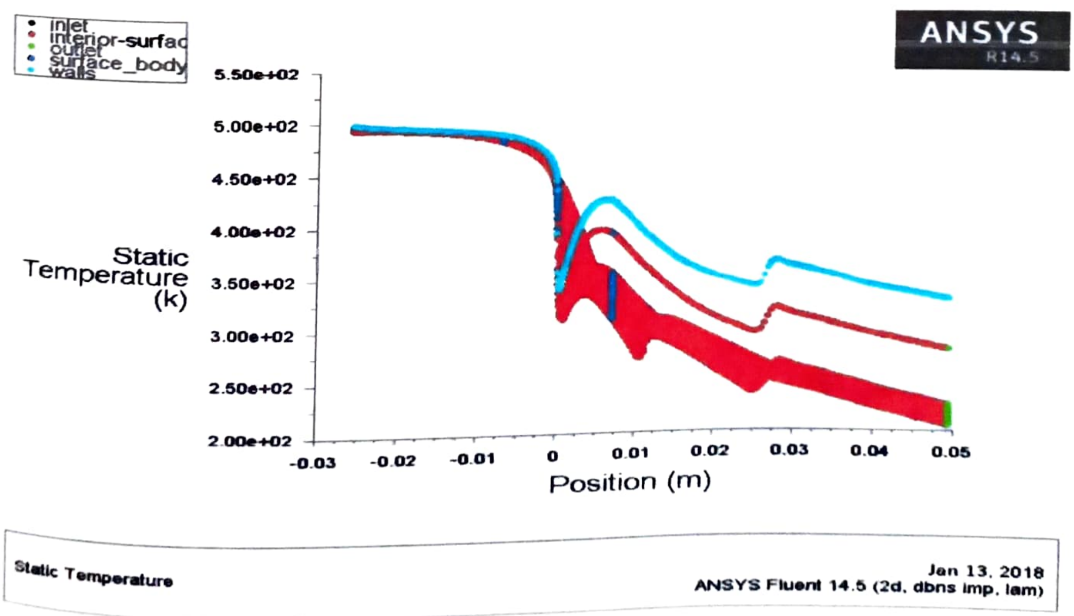


Fig 5.26 XY plot for static temperature with divergence angle of 20^0

5.5 CFD Analysis of 25° Rocket Nozzle:

The nozzle is designed as per the requirement with required dimensions, angles of inlet, outlet and walls.

5.5.1 Geometry:

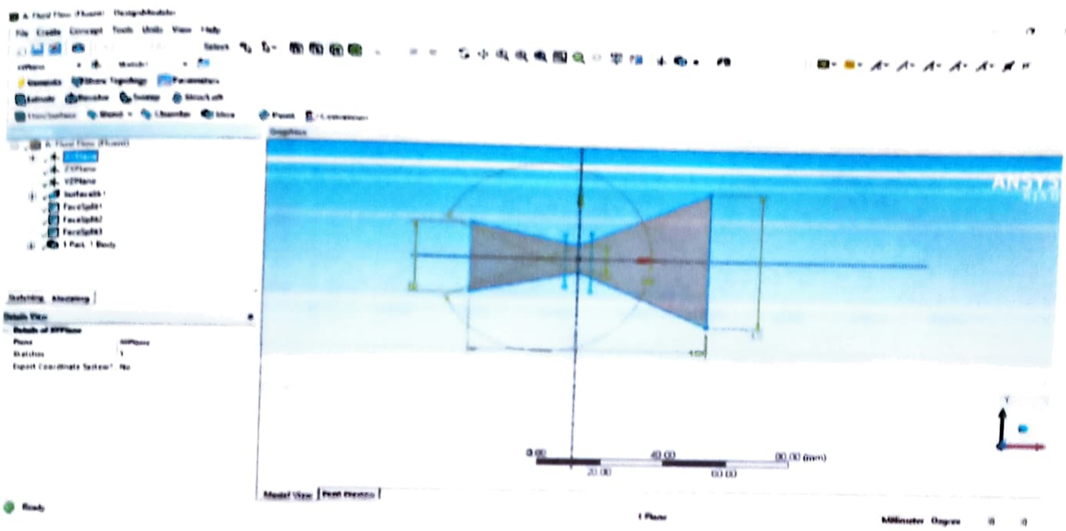


Fig 5.45 Geometry of 25^0 Rocket Nozzle

5.5.2 Meshing:

Next is obtained geometry model is meshed by using fluent software.

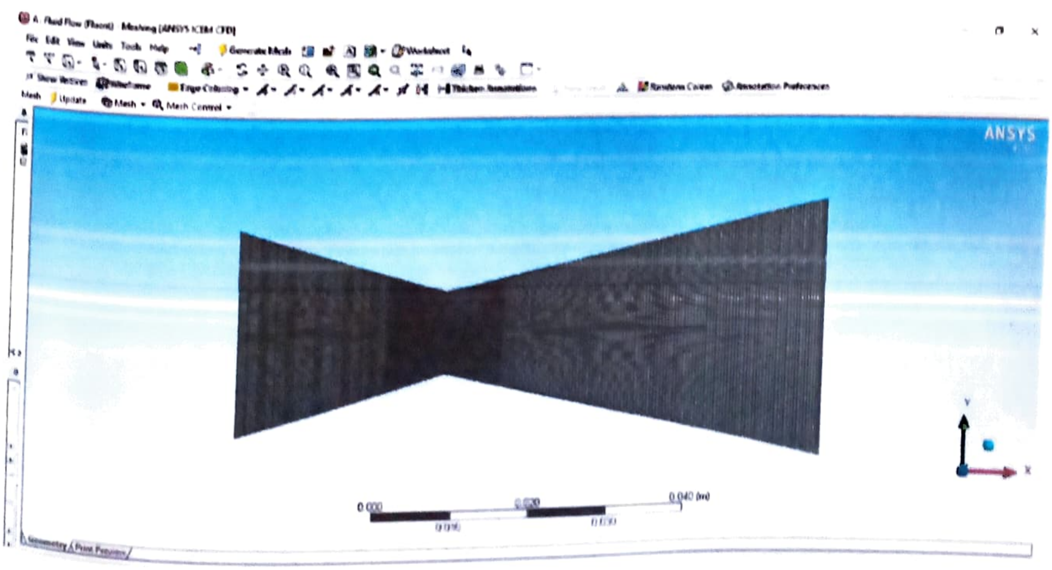


Fig 5.46 Meshed model of 25^0 nozzle

The designed rocket nozzle is made to run and solution is converged at 606 iterations.

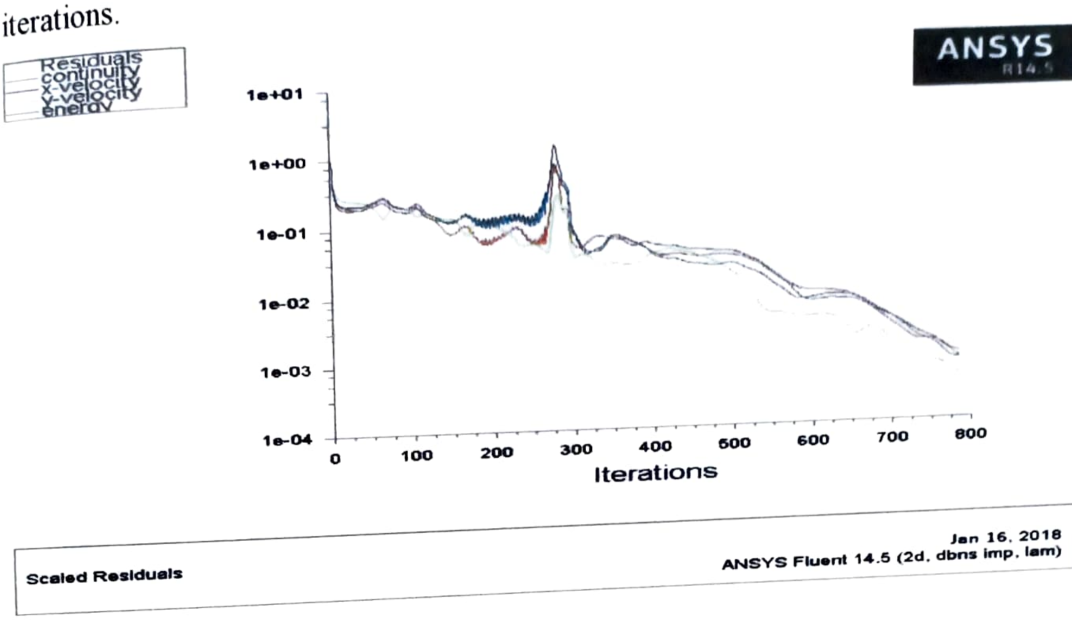


Fig 5.47 Iterations

5.5.3 Contour of Velocity:

It is clearly observed from the figure that the Mach number is 0.049 i.e. sub-sonic region in convergent section at inlet point, at the throat the Mach number is 0.97 i.e. in sonic, at the exit it becomes supersonic for the designed nozzle. Near the wall, the Mach number is 3.15. This is due to the viscosity and the turbulence in the fluid.

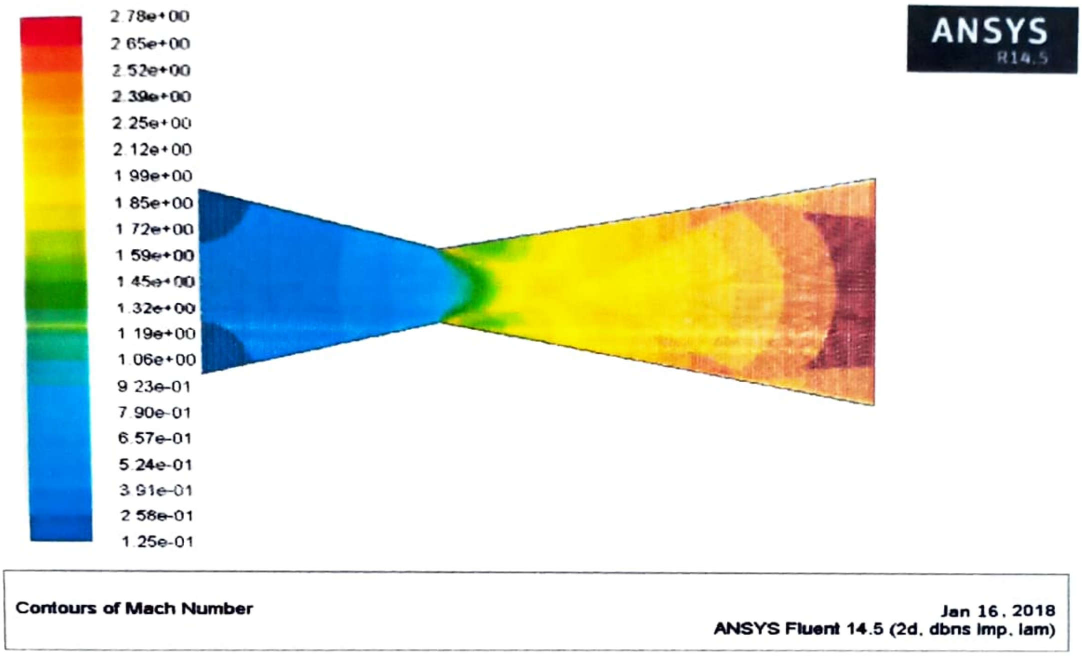


Fig 5.48 Velocity contour for Nozzle with angle 25^0

5.5.4 Pressure Contour:

Static pressure is the pressure that is exerted by a fluid. Specifically, it is the pressure measured when the fluid is still, or at rest. The above figure reveals the fact that gas gets expanded in the nozzle exit. The static pressure in the inlet is observed to be 3bar and as we move to the throat there is a decrease and the value is found to be 1.60bar. After the throat, the pressure falls in a more repaid manner towards the exit of the nozzle. At the exit it is found to be 2.94bar.

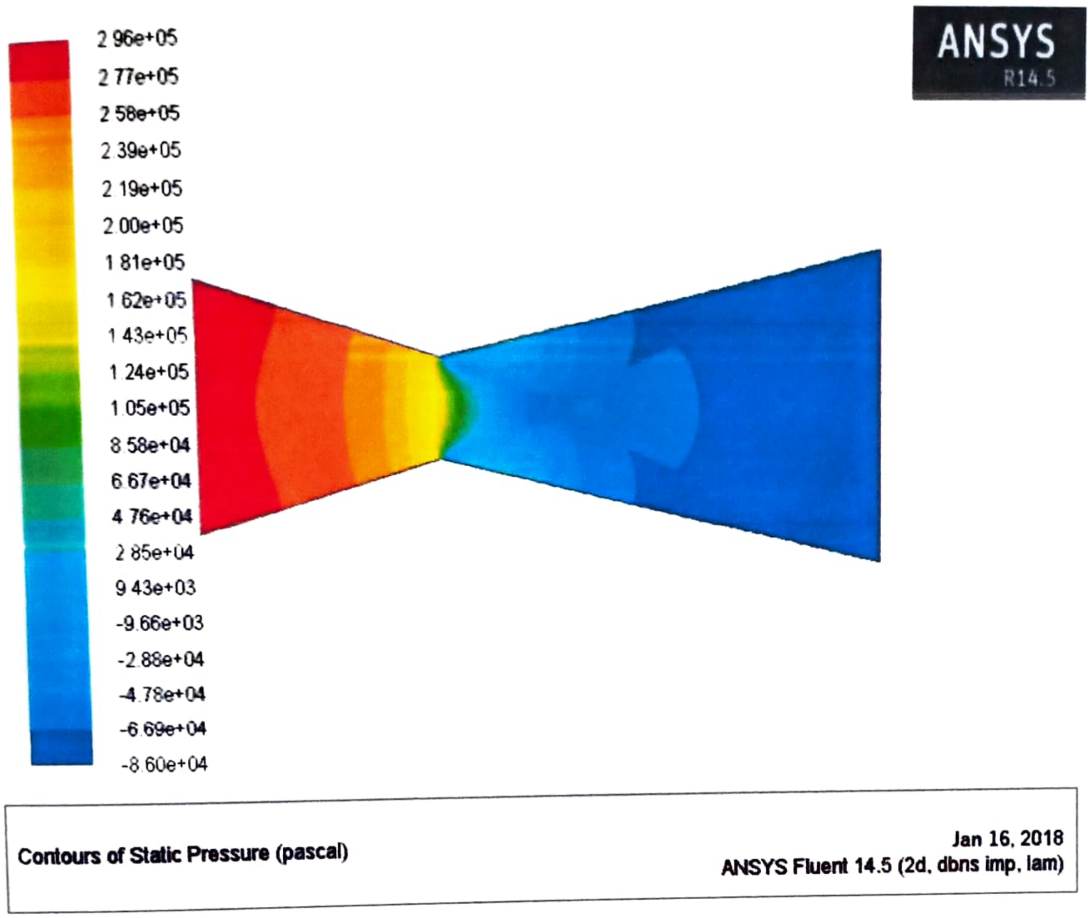


Fig 5.49 Pressure Contour for nozzle with divergence angle of 25^0

5.5.5 Temperature Contour:

The total temperature always remains a constant in the inlet up to the throat after which it tends to increase. Near the walls the temperature increases to 400 K. In the inlet and the throat the temperature is 426 K. After the throat the temperature increases to 317K at the exit.

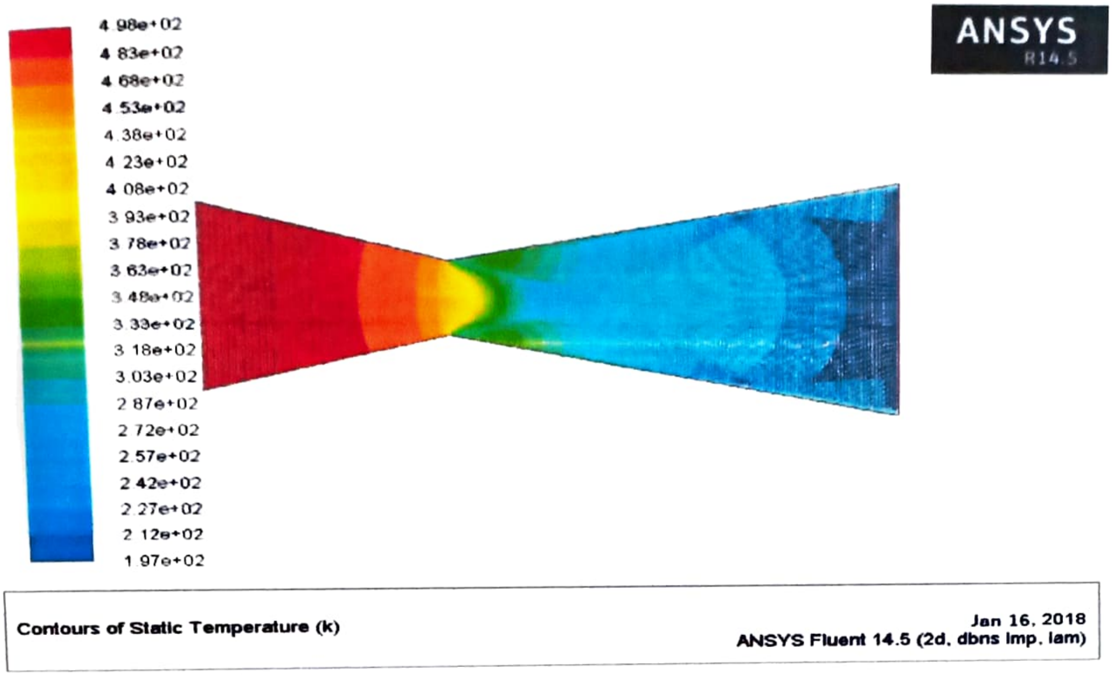


Fig 5.50 Temperature Contour for Nozzle with divergence angle 25^0

5.5.6 Velocity Plot:

A graph is plotted by taking position (m) on X-axis and Mach number on Y-axis. It is clearly observed that the velocity is increased.

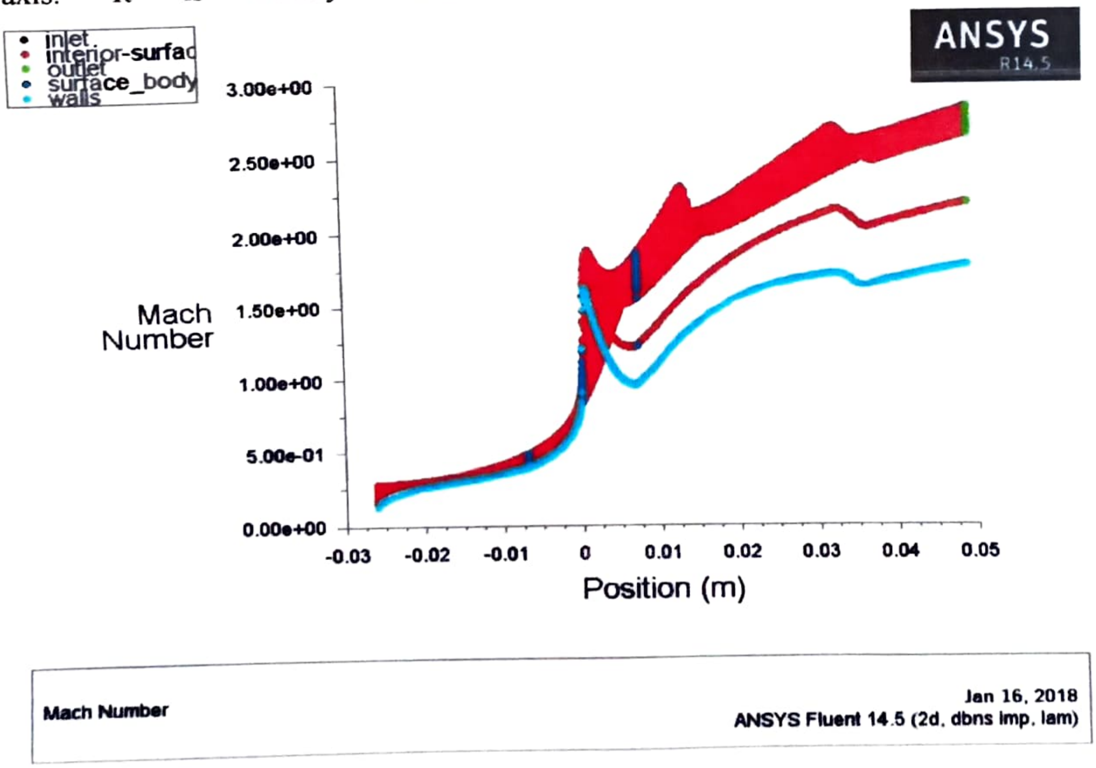


Fig 5.51 XY plot for velocity contour with divergence angle of 25^0

5.5.7 Pressure Plot:

A graph is plotted by taking position (m) on X-axis and Static Pressure (Pa) on Y-axis. There is decrease in the pressure from inlet to outlet.

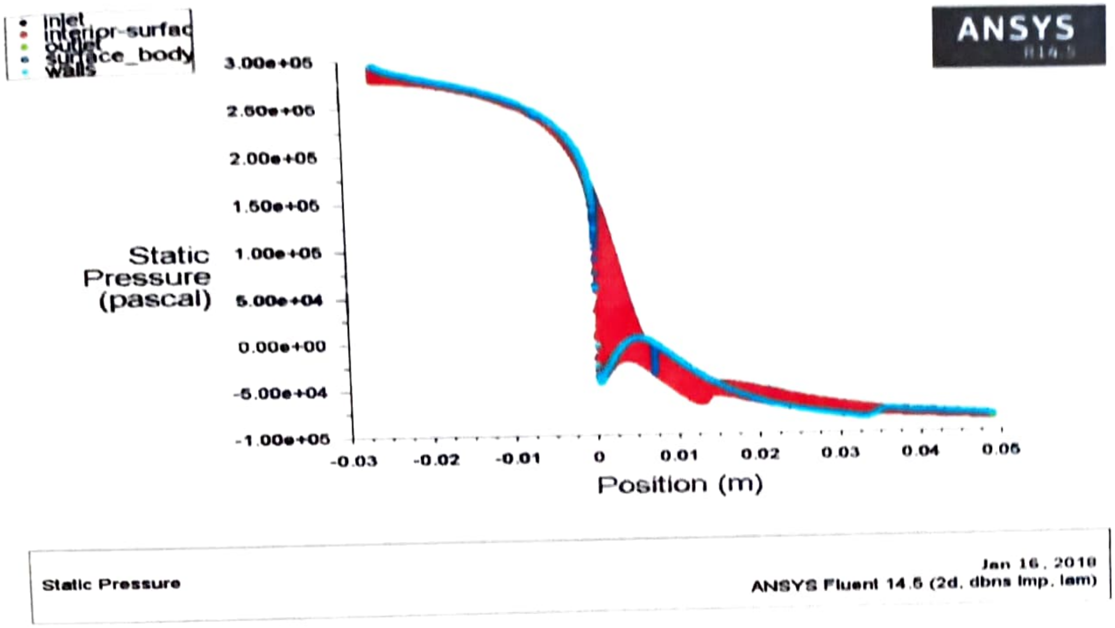


Fig 5.52 XY plot for pressure contour with divergence angle of 25^0

5.5.8 Temperature Plot:

A graph is plotted by taking position (m) on X-axis and Static temperature (k) on Y-axis. There is sudden increase and then decrease in temperature.

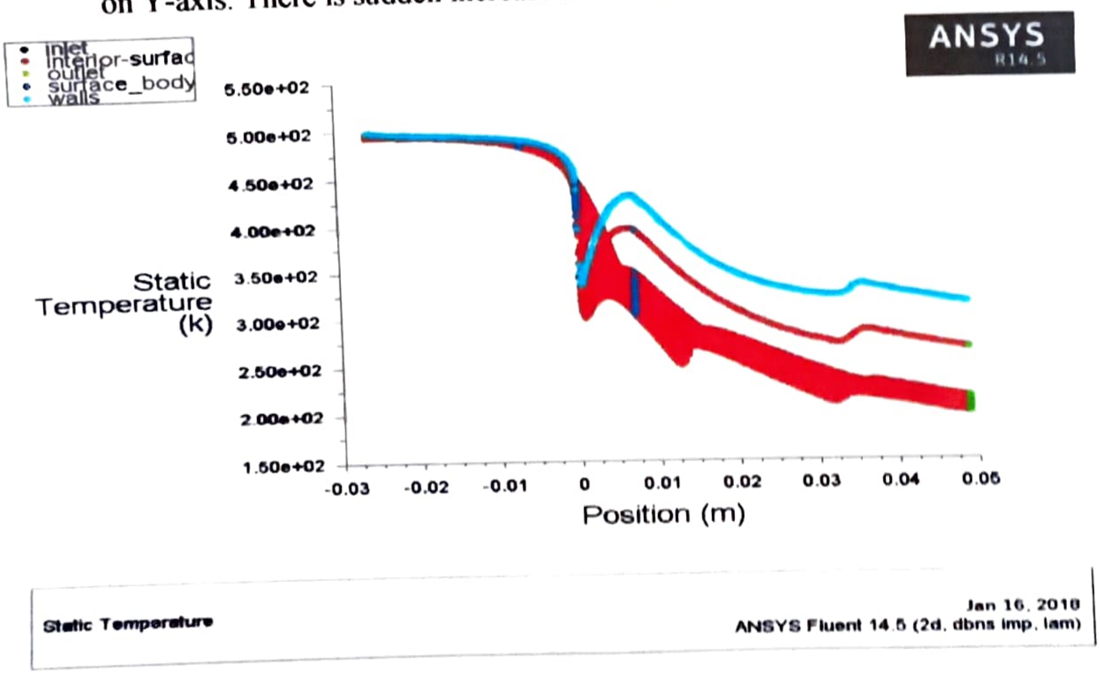


Fig 5.53 XY plot for static temperature with divergence angle of 25^0

5.6 CFD Analysis of 30^0 Rocket Nozzle:

The nozzle is designed as per the requirement with required dimensions, angles of inlet, outlet and walls.

5.6.1 Geometry:

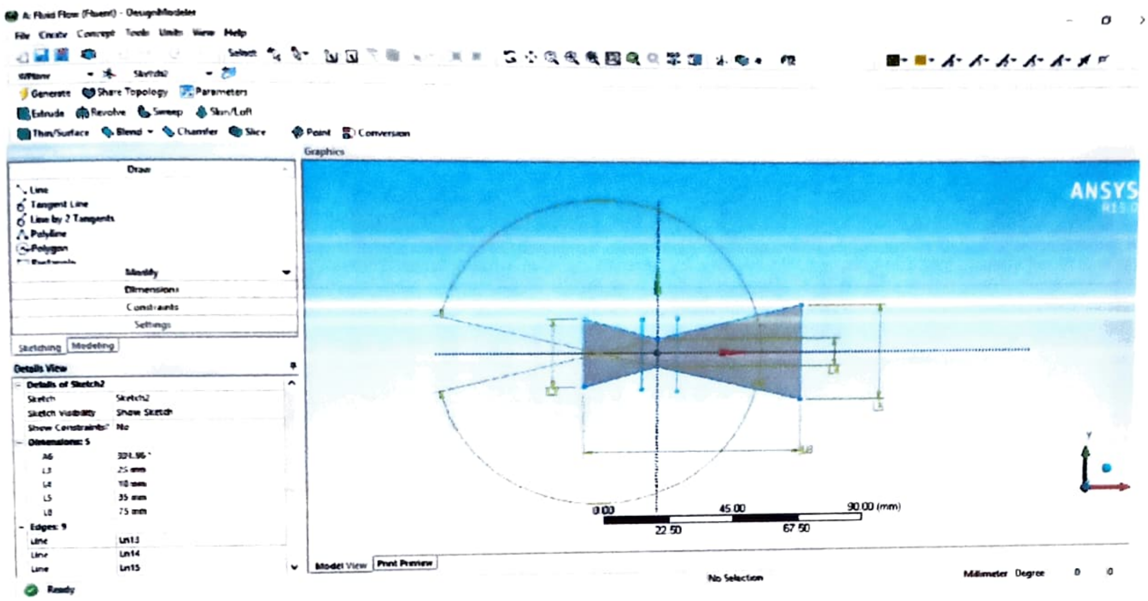


Fig 5.27 Geometry of 30^0 Rocket Nozzle

5.6.2 Meshing:

Next is obtained geometry model is meshed by using fluent software.

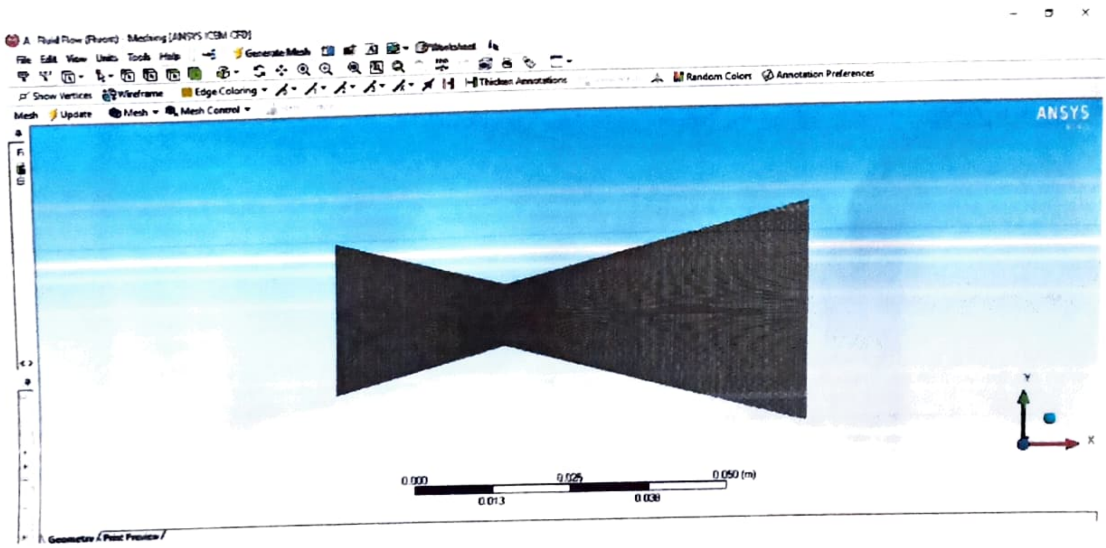


Fig 5.28 Meshed model of 30^0 nozzle

The designed rocket nozzle is made to run and solution is converged at 778 iterations.

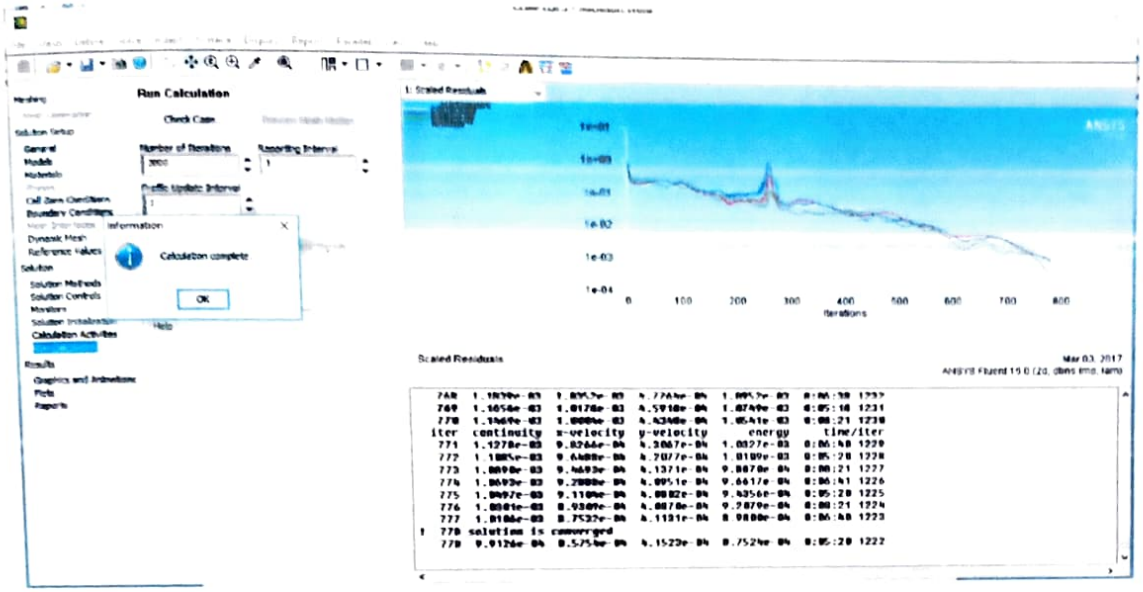


Fig 5.29 Iterations

5.6.3 Contour of Velocity:

It is clearly observed from the figure that the Mach number is 0.12 i.e., sub-sonic region in convergent section at inlet point, at the throat the Mach number is 0.81 i.e., in sonic, at the exit it becomes supersonic for the designed nozzle. Near the wall, the Mach number is 2.87. This is due to the viscosity and the turbulence in the fluid.

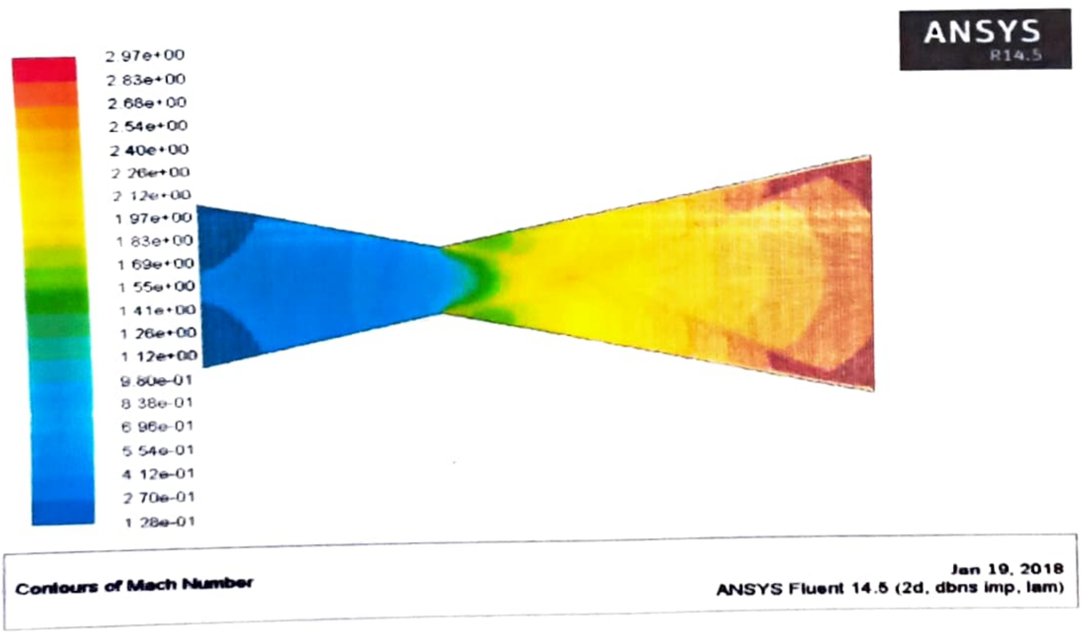


Fig 5.30 Velocity contour for Nozzle with angle 30^0

5.6.4 Pressure Contour:

Static pressure is the pressure that is exerted by a fluid. Specifically, it is the pressure measured when the fluid is still, or at rest. The above figure reveals the fact that gas gets expanded in the nozzle exit. The static pressure in the inlet is observed to be 3bar and as we move to the throat there is a decrease and the value is found to be 1.56bar. After the throat, the pressure falls in a more repaid manner towards the exit of the nozzle. At the exit it is found to be 2.96bar.

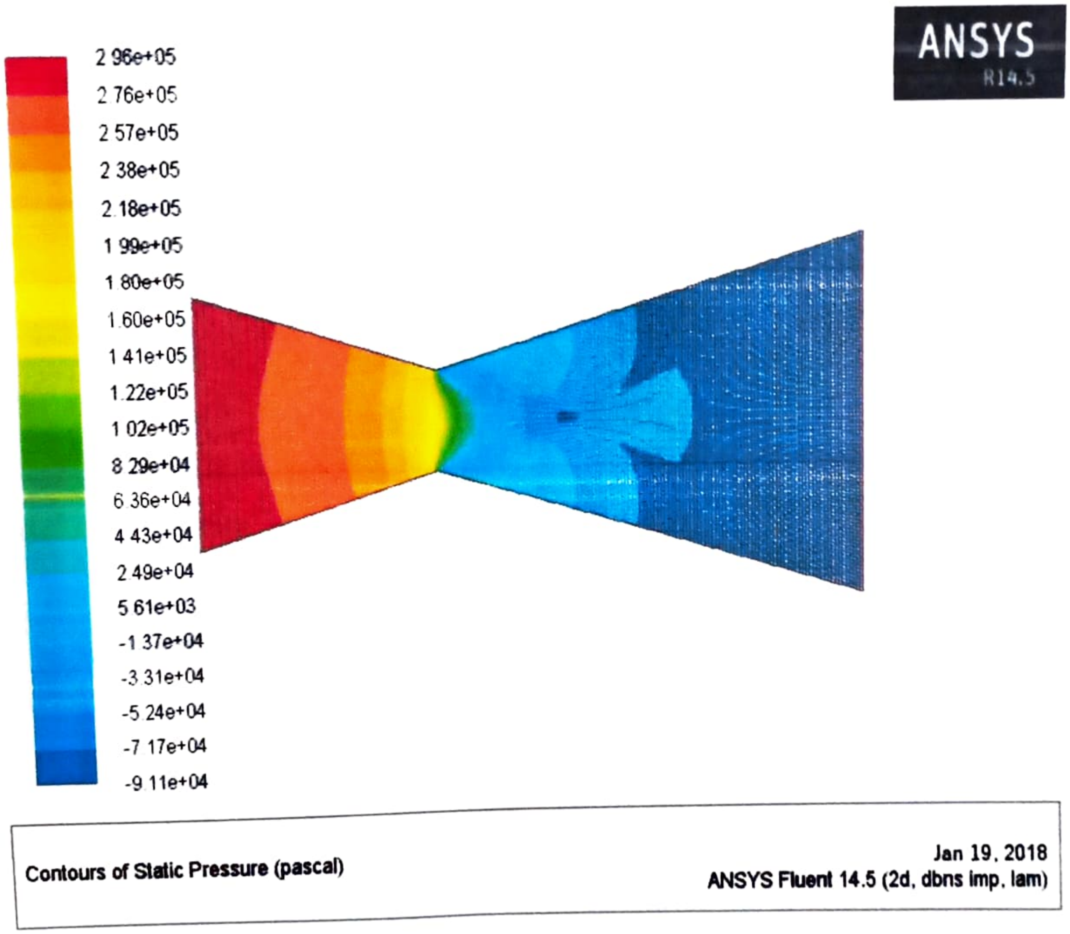


Fig 5.31 Pressure Contour for nozzle with divergence angle of 30^0

5.6.5 Temperature Contour:

The total temperature always remains a constant in the inlet up to the throat after which it tends to increase. Near the walls the temperature increases to 400K. In the inlet and the throat the temperature is 337K. After the throat the temperature increases to 449K at the exit.

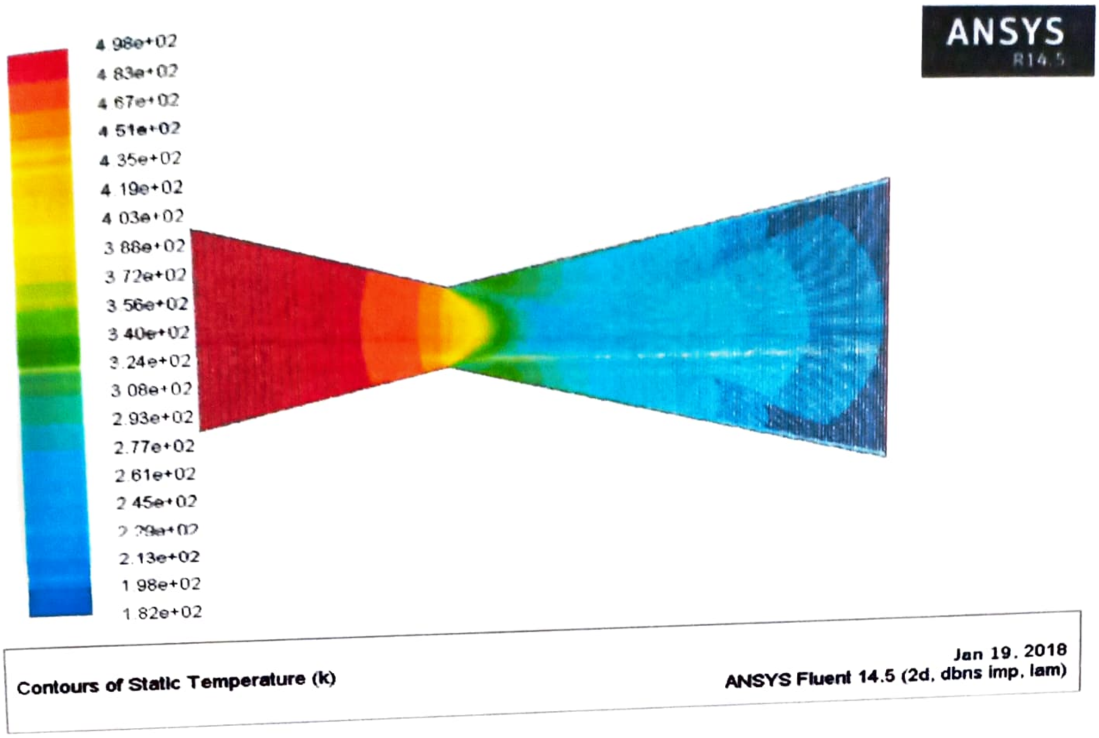


Fig 5.32 Temperature Contour for Nozzle with divergence angle 30^0

5.6.6 Velocity Plot:

A graph is plotted by taking position (m) on X-axis and Mach number on Y-axis. It is clearly observed that the velocity is increased.

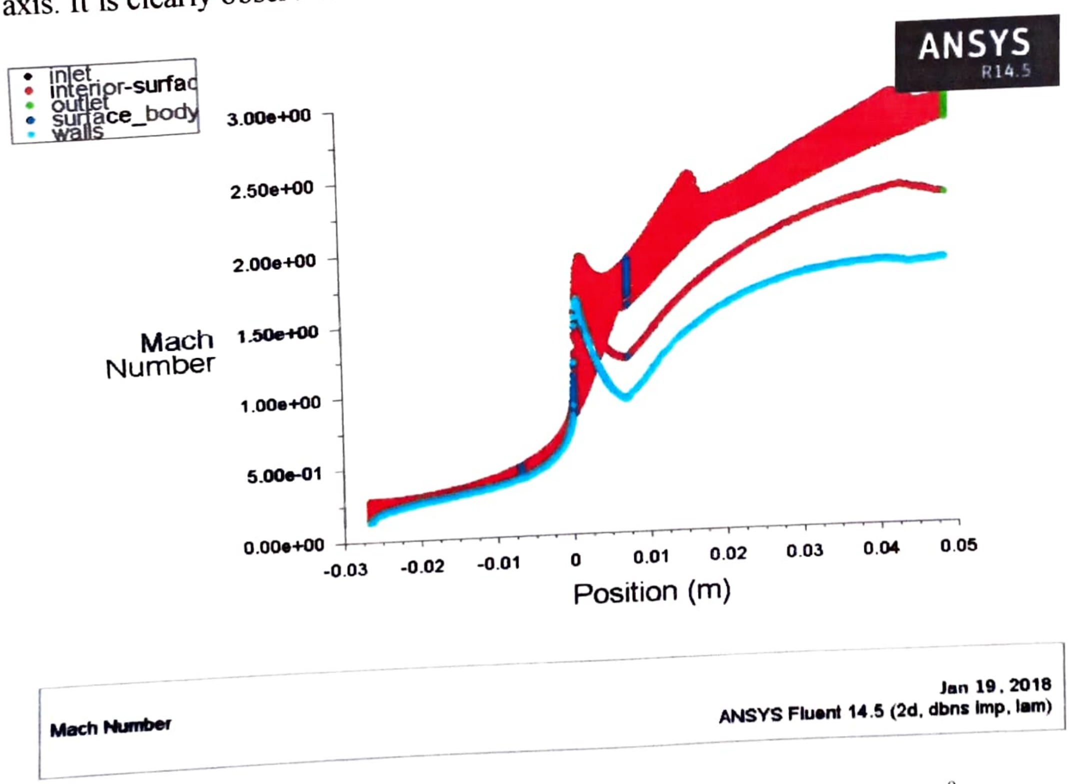


Fig 5.33 XY plot for velocity contour with divergence angle of 30^0

5.6.7 Pressure Plot:

A graph is plotted by taking position (m) on X-axis and Static Pressure (Pa) on Y-axis. There is decrease in the pressure from inlet to outlet.

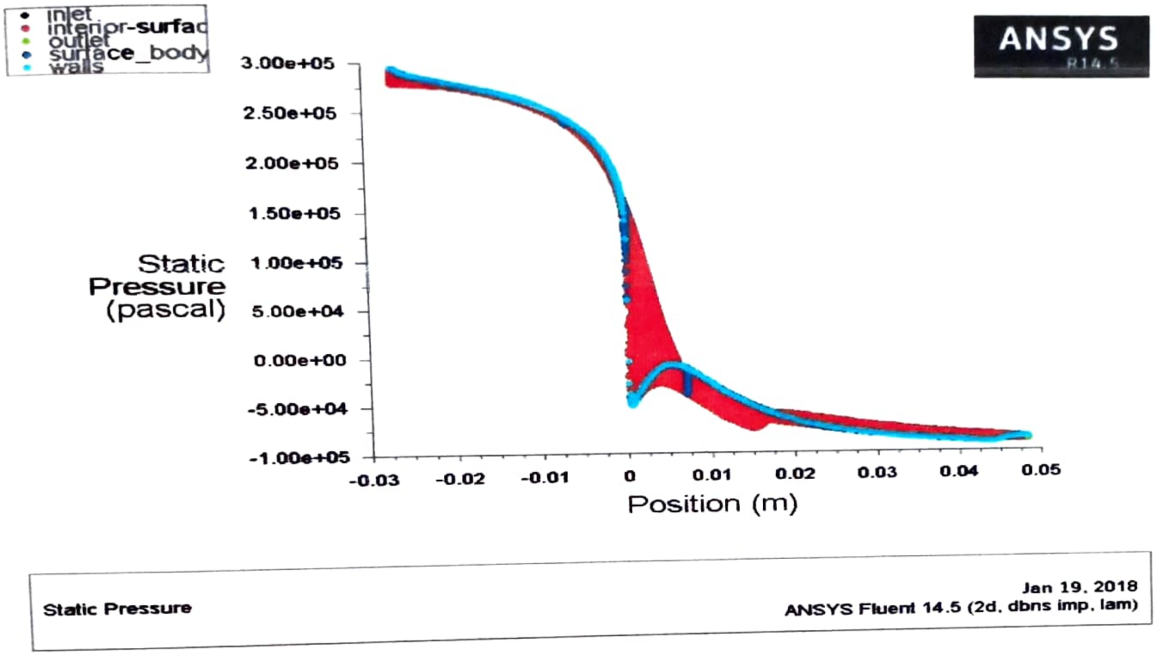


Fig 5.34 XY plot for pressure contour with divergence angle of 30^0

5.6.8 Temperature Plot:

A graph is plotted by taking position (m) on X-axis and Static temperature (k) on Y-axis. There is sudden increase and then decrease in temperature.

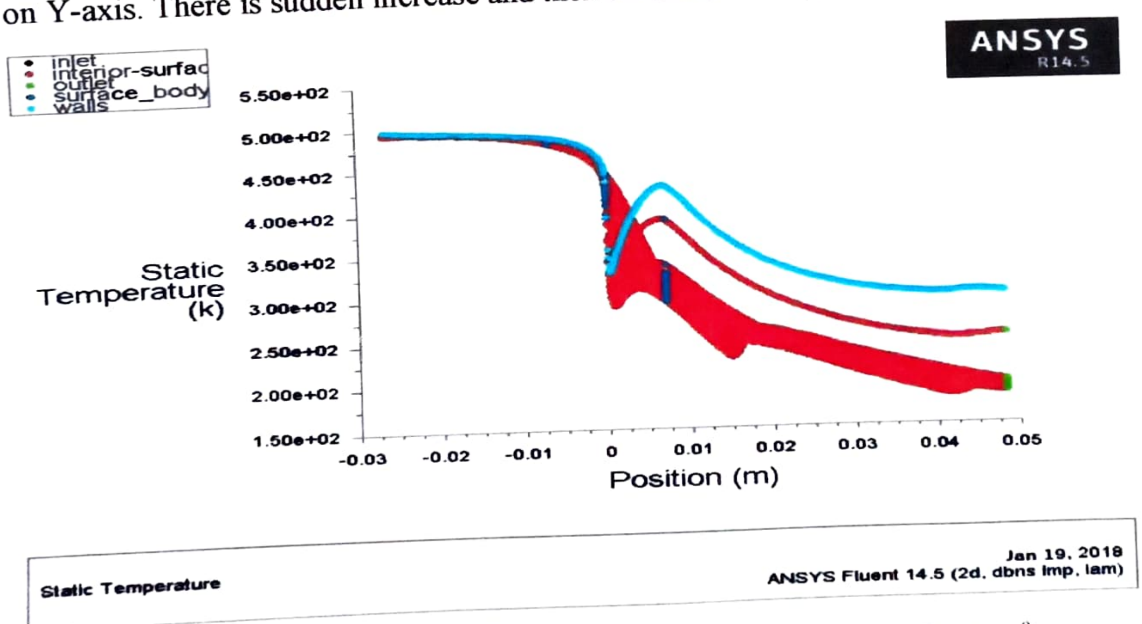


Fig 5.35 XY plot for static temperature with divergence angle of 30^0

5.7 CFD Analysis of 35° Rocket Nozzle:

The nozzle is designed as per the requirement with required dimensions, angles of inlet, outlet and walls.

5.7.1 Geometry:

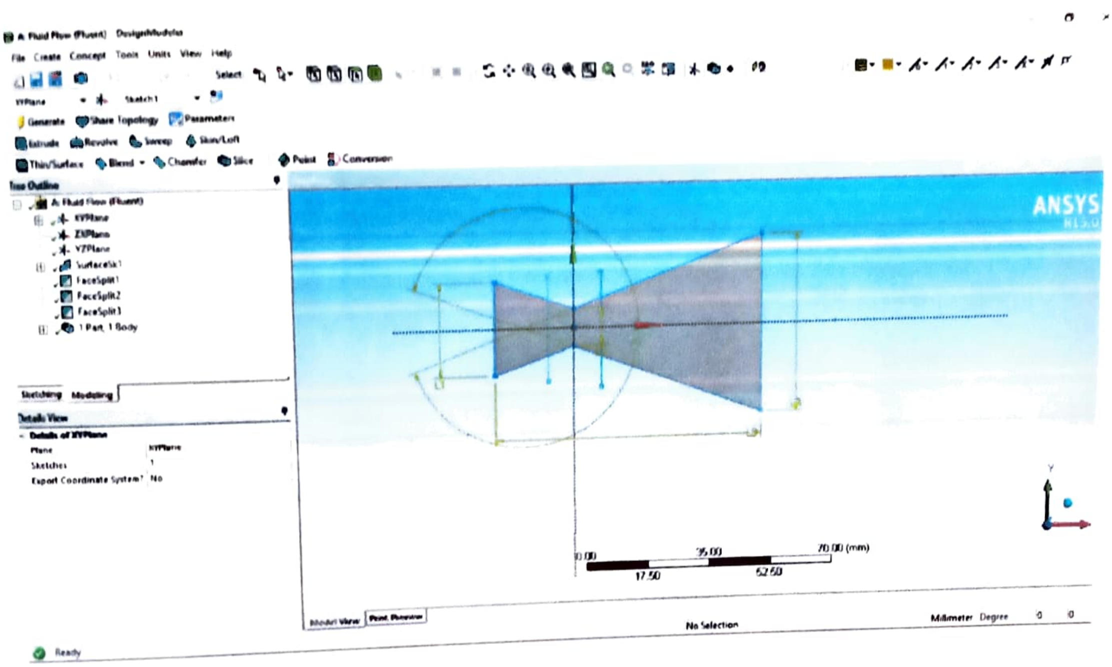


Fig 5.36 Geometry of 35^0 Rocket Nozzle

5.7.2 Meshing:

Next is obtained geometry model is meshed by using fluent software.

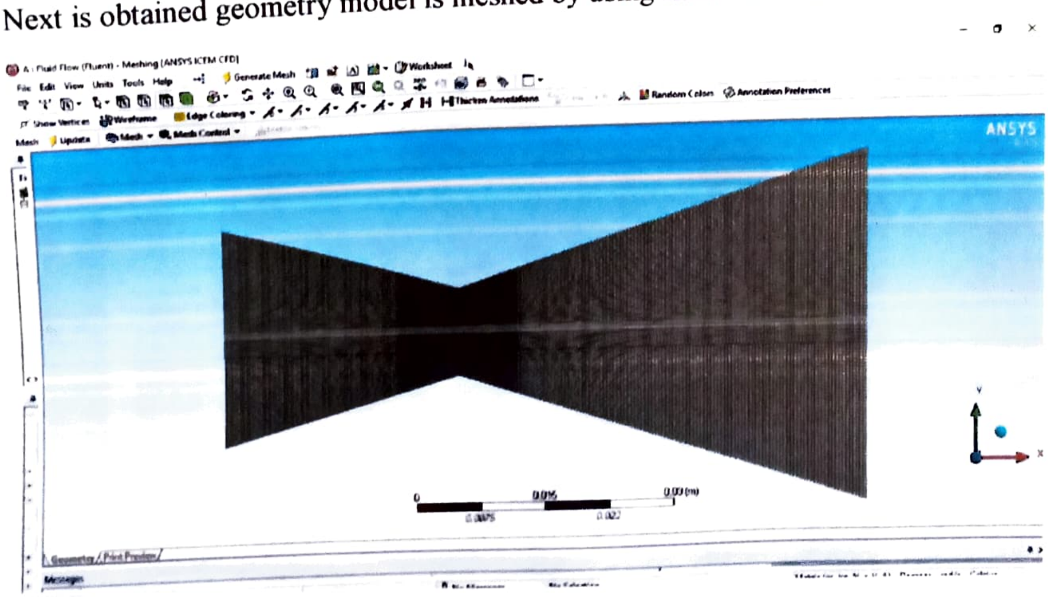


Fig 5.37 Meshed model of 35^0 nozzle

The designed rocket nozzle is made to run and solution is converged at 581 iteration.

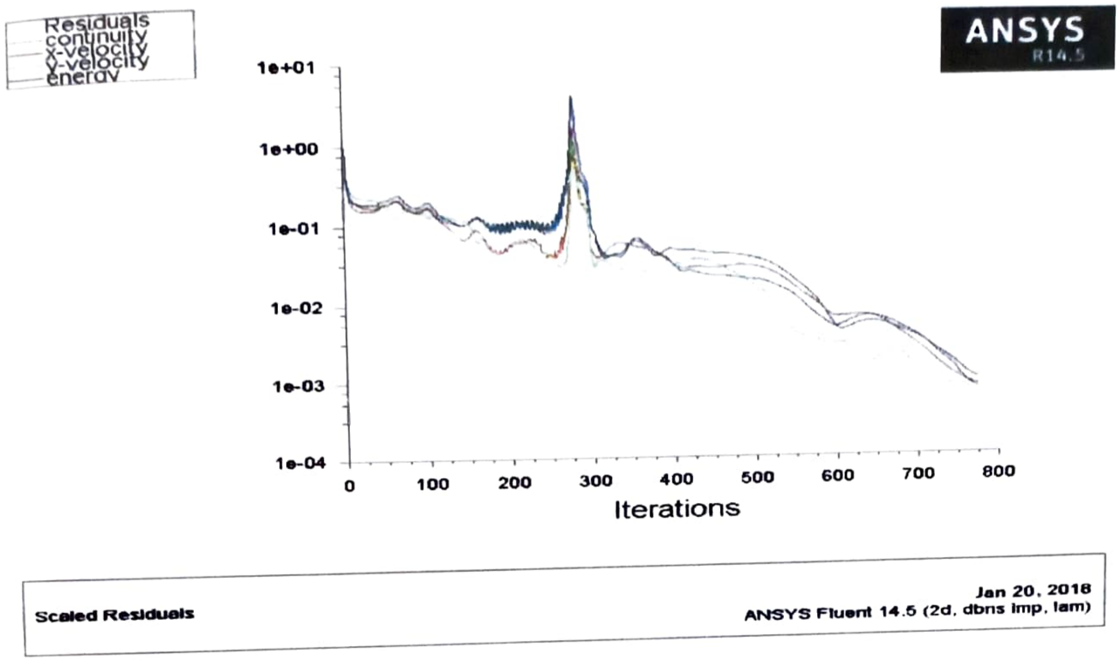


Fig 5.38 Iterations

5.7.3 Contour of Velocity:

It is clearly observed from the figure that the Mach number is 0.10 i.e. sub-sonic region in convergent section at inlet point, at the throat the Mach number is 0.92 i.e. in sonic, at the exit it becomes supersonic for the designed nozzle. Near the wall, the Mach number is 3.37. This is due to the viscosity and the turbulence in the fluid.

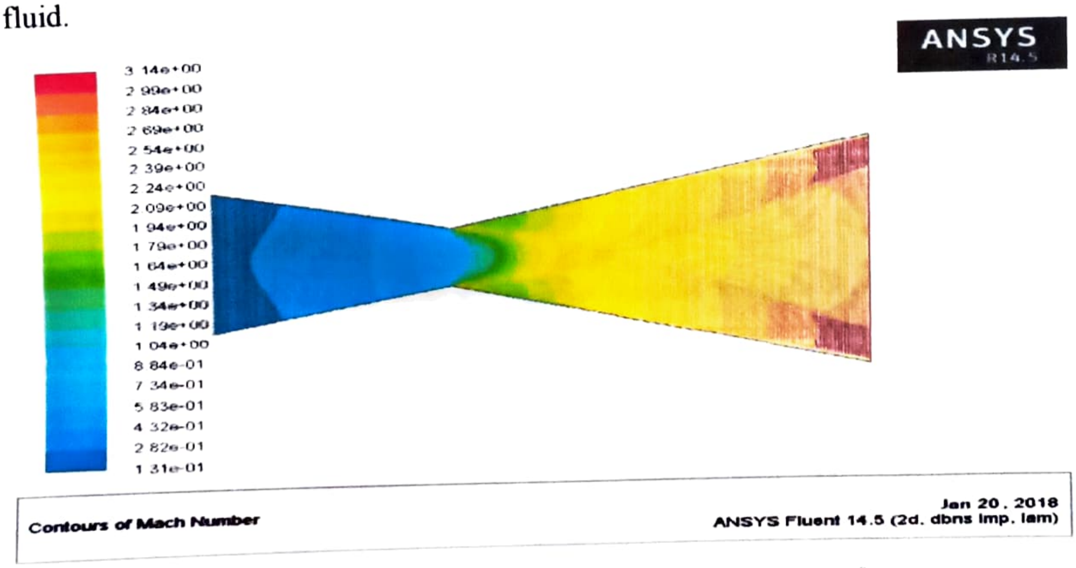


Fig 5.39 Velocity contour for Nozzle with angle 35^0

5.7.4 Pressure Contour:

Static pressure is the pressure that is exerted by a fluid. Specifically, it is the pressure measured when the fluid is still, or at rest. The above figure reveals the fact that gas gets expanded in the nozzle exit. The static pressure in the inlet is observed to be 3bar and as we move to the throat there is a decrease and the value is found to be 1.61bar. After the throat, the pressure falls in a more repaid manner towards the exit of the nozzle. At the exit it is found to be 2.97bar.

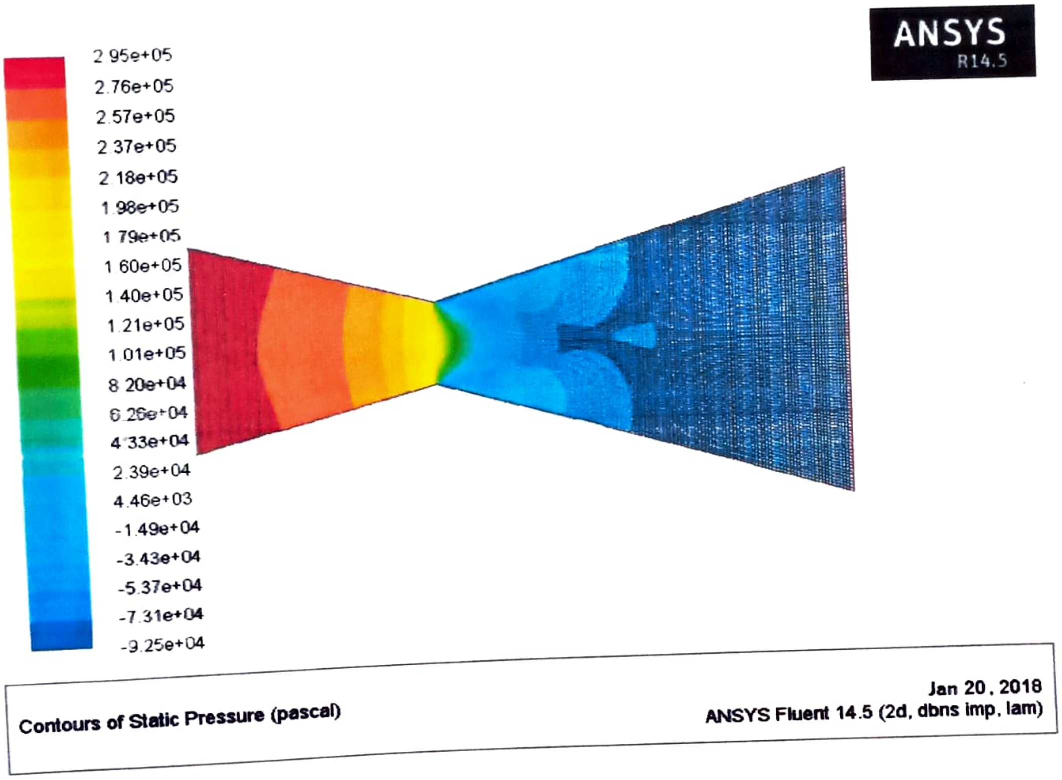


Fig 5.40 Pressure Contour for nozzle with divergence angle of 35^0

5.7.5 Temperature Contour:

The total temperature always remains a constant in the inlet up to the throat after which it tends to increase. Near the wall temperature increases to 400K. In the inlet and the throat the temperature is 317K. After the throat the temperature increases to 399.1K at the exit.

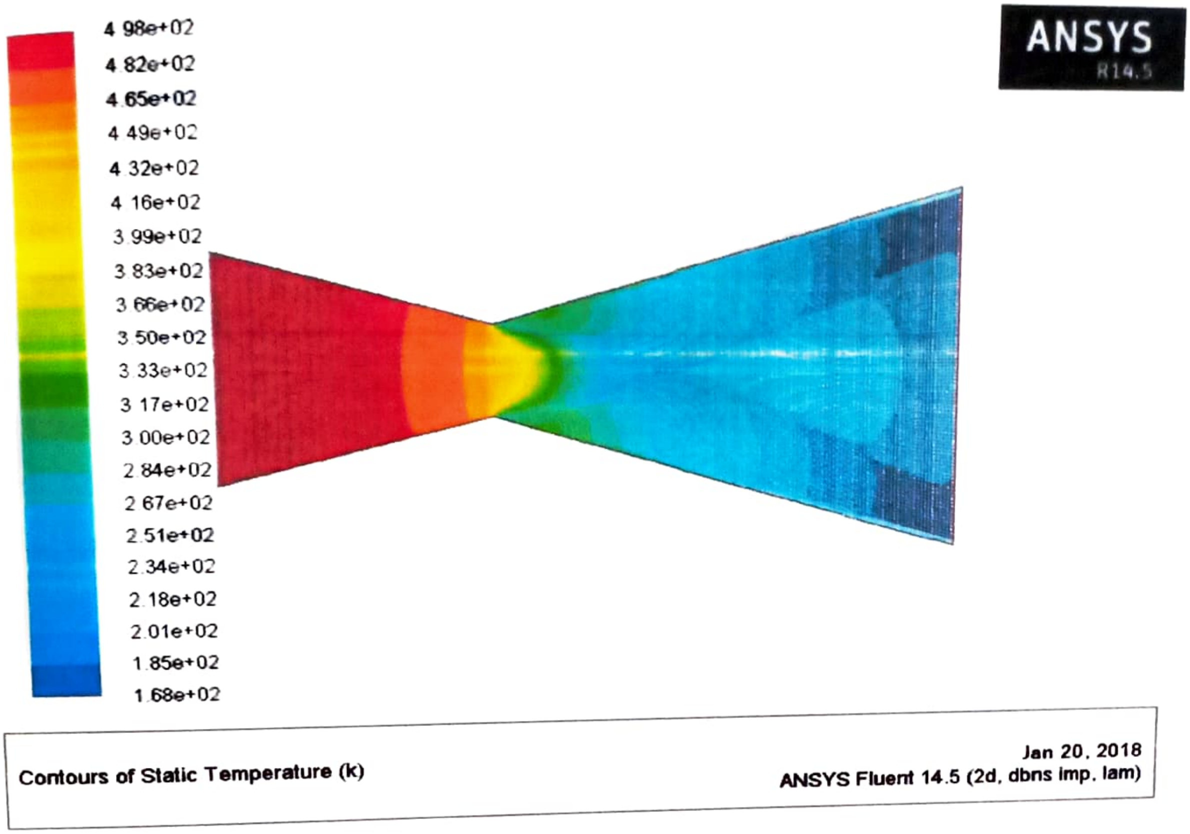


Fig 5.41 Temperature Contour for Nozzle with divergence angle 35^0

5.7.6 Velocity Plot:

A graph is plotted by taking position (m) on X-axis and Mach number on Y-axis. It is clearly observed that the velocity is increased.

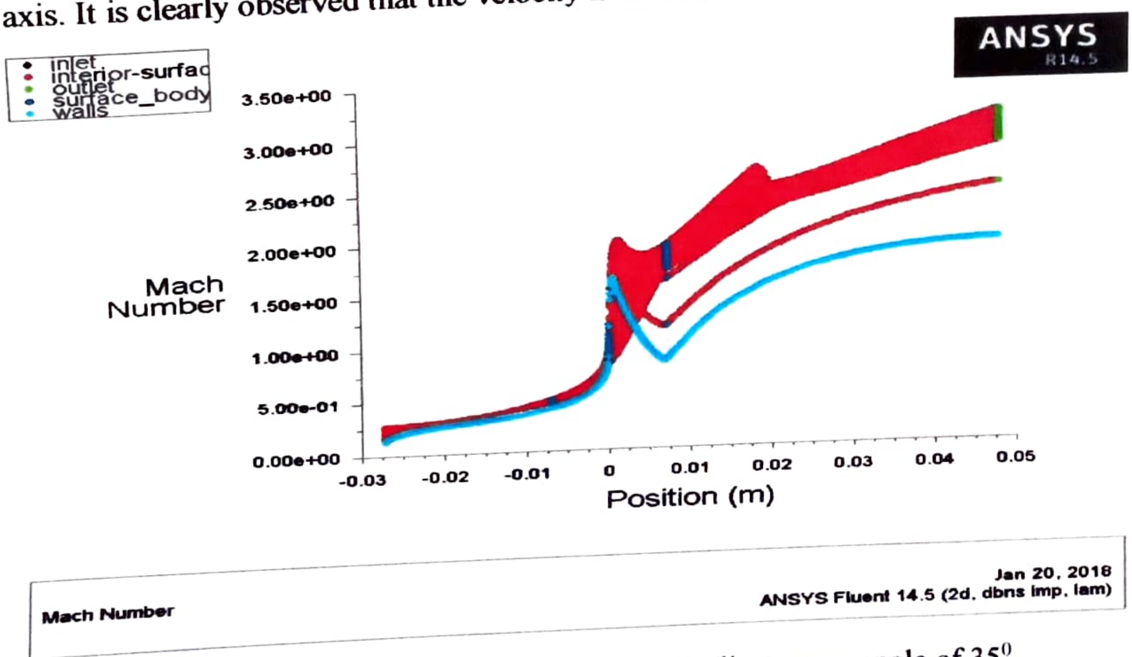


Fig 5.42 XY plot for velocity contour with divergence angle of 35^0

5.7.7 Pressure Plot:

A graph is plotted by taking position (m) on X-axis and Static Pressure (Pa) on Y-axis. There is decrease in the pressure from inlet to outlet.

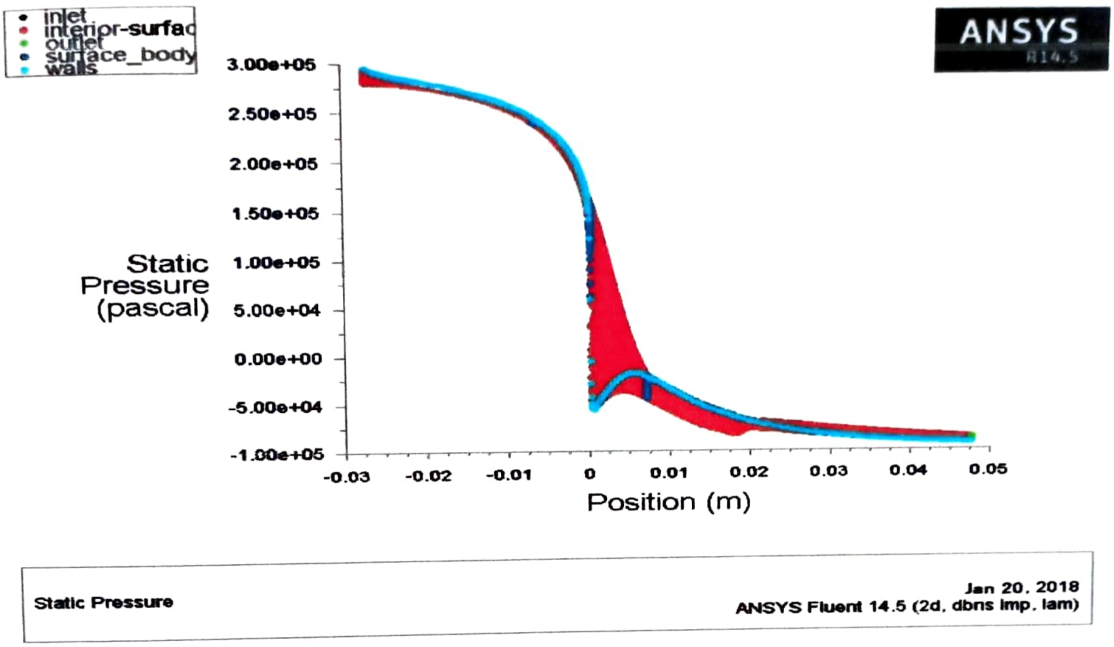


Fig 5.43 XY plot for pressure contour with divergence angle of 35^0

5.7.8 Temperature Plot:

A graph is plotted by taking position (m) on X-axis and Static temperature (k) on Y-axis. There is sudden increase and then decrease in temperature.

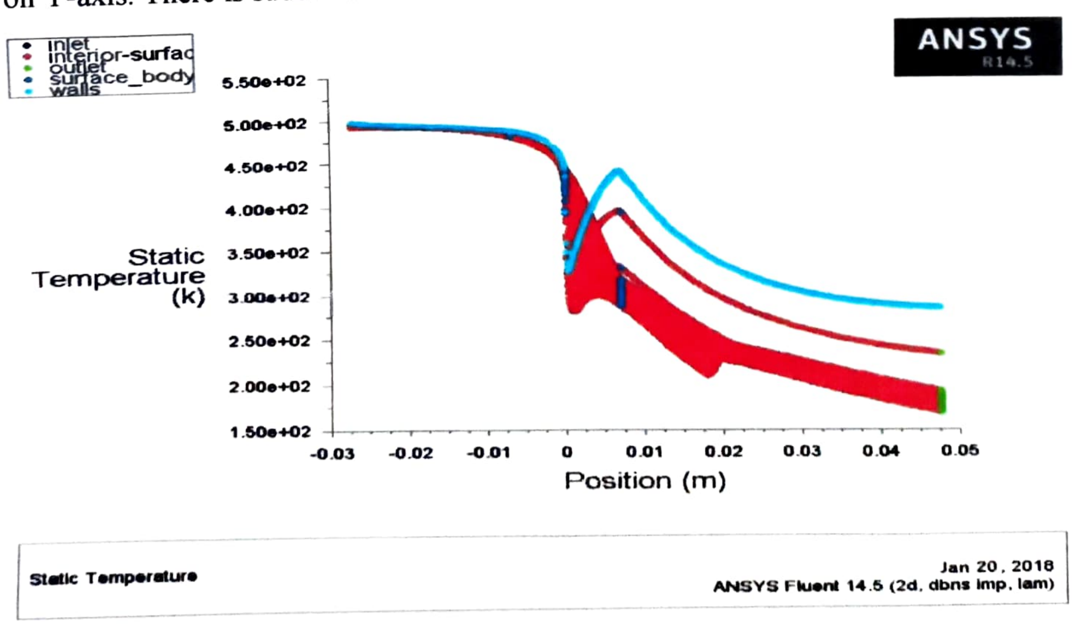


Fig 5.44 XY plot for static temperature with divergence angle of 35^0

6 RESULTS AND DISCUSSION

Conditions at Exit:

Table 6.1 Conditions at Exit

Case	Angle	Mach Number
1	10	2.20e+00
2	15	2.45e+00
3	20	2.70e+00
4	25	2.78e+00
5	30	2.97e+00
6	35	3.14e+00

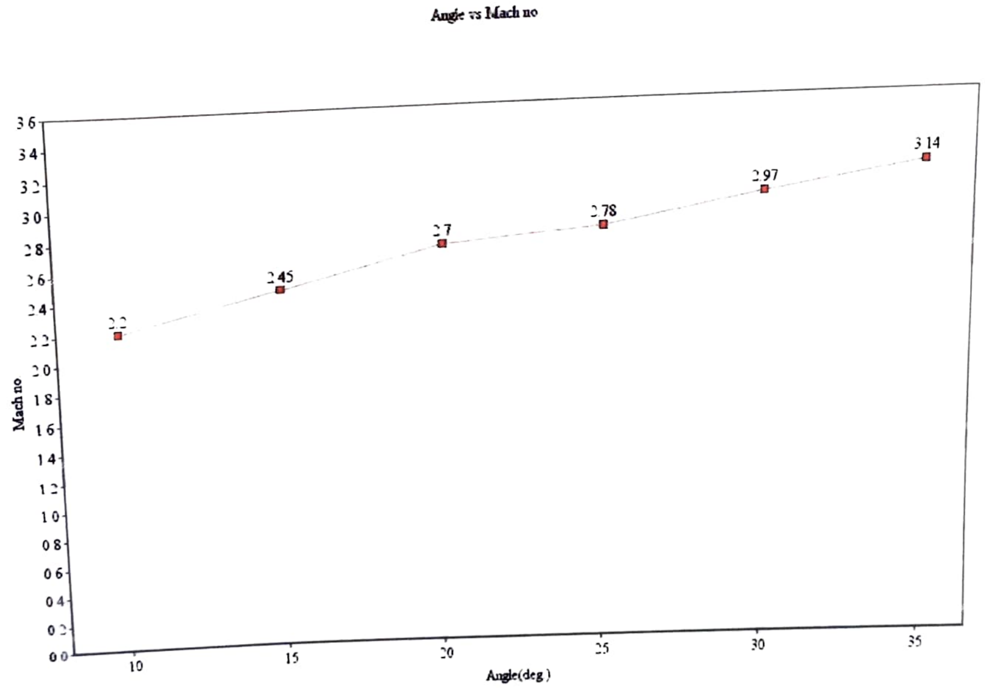


Fig 6.1 Comparison of Mach numbers at different divergence angles at exit.

Conditions at Throat:

Table 6.2 conditions at Throat

Case	Angle	Mach Number
1	10	0.639
2	15	0.819
3	20	0.896
4	25	0.923
5	30	0.980
6	35	1.040

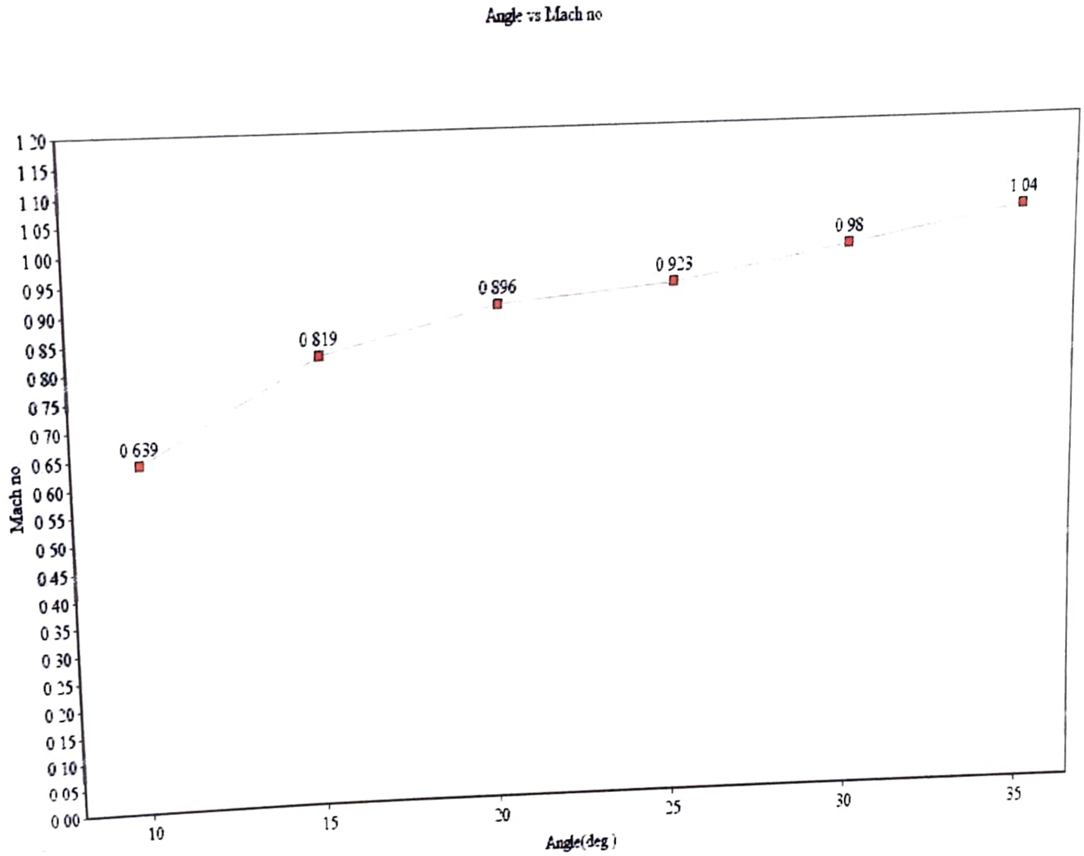


Fig 6.2 Comparison of Mach number at different divergence angles at throat.

7 CONCULUSION

- A model was developed to determine the pressure, temperature and velocity distribution (in terms of Mach number) from inlet to the exit of the rocket nozzle. A study is carried out by varying the divergence angles. The divergence angles considered for the study of flow are 10^0, 15^0, 20^0, 25^0, 30^0 and 35^0.
- From the literature review nozzle dimensions are considered and sufficient boundary conditions are implemented.
- The following observations were found in the nozzle with different divergence angles considering the default divergence angle as 10^0.
    - In the nozzle when the divergence angle is 35^0, Mach number is 1.04 at throat and at the divergence angle of 10^0 the Mach number is 0.639.
    - At the throat the velocity magnitude is same for all divergence angles.
    - When the divergence angle is 10^0, Mach number at the nozzle exit is 2.20 while the Mach number at the nozzle exit is 3.14 at the nozzle exit for the divergence angle of 35^0.
- The efficiency increases with the increase in the divergence angle up to 35^0.

References

- **Paradhasaradhi Natta** [1] - Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (Cfd), Vol. 2, Issue 5, September- October 2012

- **K.M.Pandey, S.K.Yadav** [2] - CFD Analysis of a Rocket Nozzle with FourInlets at Mach 2.1, Volume 1, Number 4, Dec 2010

- **B.V.V.Naga Sudhakar** .[3]- Modeling and simulation of Convergent-Divergent Nozzle Using Computational Fluid Dynamics, Vol. 03, Issue. 08, Aug-2016

- **K.P.S.Surya Narayana, K.Sadhasiva Reddy** [4]- Simulation of Convergent Divergent Rocket Nozzle using CFD Analysis, Volume 13, Issue 4 Ver. I, Jul. - Aug. 2016

- **Gutte Rajeshwara Rao [5]** - Flow Analysis in a Convergent-Divergent Nozzle Using CFD, Vol. 1, Issue. 2, Oct-2013

- **Nikhil D. Deshpande** [6] - Theoretical & CFD Analysis Of De Laval Nozzle, International Journal of Mechanical and Production Engineering (IJMPE) , pp. 33-36, Volume-2,Issue-4, Apr-2014

- **Dr.Y.V.Hanumantha Rao.** [7] - Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (Cfd), Vol. 2, Issue 5, September- October-2012

- **Biju Kuttan. P,M. Sajesh.**[8] -Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics, Volume 2, Issue 2, 2013

- **Prosun Roy, Abhijit Mondal, Biswanath Barai**.[9]- CFD Analysis Of Rocket Engine Nozzle, Vol-3, Issue-1 , Jan- 2016