Computational Analysis of Fluid Flow in a Rocket Nozzle

By

Muhammad Waqas Khalid

School of Chemical and Materials Engineering (SCME)

National University of Sciences and Technology (NUST)

2019

Computational Analysis of Fluid Flow in a Rocket Nozzle

Names: Muhammad Waqas Khalid
Reg.No:202998

This thesis is submitted as a partial fulfillment of the requirements for the degree of

MS in Energetic Materials Engineering

Supervisor Name: Dr. Muhammad Ahsan

School of Chemical and Materials Engineering (SCME)
National University of Sciences and Technology (NUST), H-12
Islamabad, Pakistan

August, 2019

Dedication

Dedicated to my Beloved Parents, wife and family.

Acknowledgments

All acclaim and eminence be to "ALLAH" a definitive creator of this universe, who endowed us with the ability to comprehend and made us curious to investigate this entire universe. Infinite greetings upon the leader of this universe and hereafter "HOLY PROPHET HAZRAT MUHAMMAD (P.B.U.H)": the wellspring of beneficial information and blessings for whole humankind and Uma.

My deepest thanks to Principal SCME Dr. Arshad Hussain for being a source of great enthusiasm and for providing all facilities of this research work.

I am highly thankful to my respected supervisor, Dr. Muhammad Ahsan for the guidance and assistance during this project. It is his consistent motivation that makes me able to achieve this land mark. His guidance helped me in all phases of research starting from learning of tools till final composing of this thesis.

I offer my deepest thanks to Dr. Abdul Qadeer Malik for his kind help and support as GEC member. It was my honor to be his student. I am also grateful to Dr. Sarah Farrukh, for efficient help during this project. It was her who taught me the basics to start the research work and finally I have completed my work. I additionally put on record, my feeling of gratitude for Lt. Col (R) Nadeem Ehsan who has always guided , supported and helped me during my MS degree at SCME. I additionally thankful to everyone, who straightforwardly or by implication, helped me to finish this work.

My sincere thanks also goes to Mr. Gulfam and his team for making the simulation lab available whenever it was required during the research phase.

I had a memorable time of 2 years during my MS degree with my seniors, Wg Cdr Nauman Sahu and Sqn Ldr Muhammad Muzammil. I wish them all the best wishes for future assignments.

I am highly obliged to my family for their eternal love. Thanks for supporting me on each and every step of my research work. It was not possible without your support.

Abstract

The thrust produced by a rocket motor is largely dependent upon the expansion of the product gases through a nozzle. The nozzle is used to accelerate the gases produced in the combustion chamber and convert the chemical-potential energy into kinetic energy so that the gases exit the nozzle at very high velocity. It converts the high pressure, high temperature and low velocity gas in the combustion chamber into high velocity gas of lower pressure and low temperature. The design of a nozzle has special importance in determining the thrust and performance of a rocket. In recent years, the design of the nozzle has received considerable attention as it directly impacts the overall performance of the rocket. This study aims to analyze the variation of flow parameters like pressure, temperature and velocity using finite volume method (FVM) solver with the standard k-ε turbulence model in computational fluid dynamics (CFD). The simulation of shockwave inside the divergent nozzle section through CFD is also investigated. In this regard, 06 nozzles have been designed using Design Modeler and CFD analysis of flow through the nozzles has been carried out using ANSYS Fluent. Three different cases were applied on all the design to investigate the performance. Based on the standard performance parameters available in the literature, one nozzle with the better performance was selected. The best design was further validated with already existing design in the literature by giving the same boundary conditions. This study's design has shown better performance in all parameters compared to the design available in the literature.

Keywords: Converging-Diverging (C-D) Nozzle; CFD; Fluent; Shockwave; standard k-ε model

Table of Contents

List of Figures

List of Tables

Chapter 1

Introduction

1.1 Background

The nozzle is widely used in number of different places, from rocket propulsion systems to very basic fuel sprayer, the nozzle application is very evident in industrial, aerospace, automobile and number of other sectors. The nozzle is a major part of missile or rocket system. Missile is a self-guided and self-propelled flying weapon. There are various types of missile as shown in Figure 1. The surface launched missiles are comparatively large in size and are designed to hit the aerial target or ground target. Their basic purpose defines the amount of thrust required during the flight which will further define the type and configuration of nozzle.

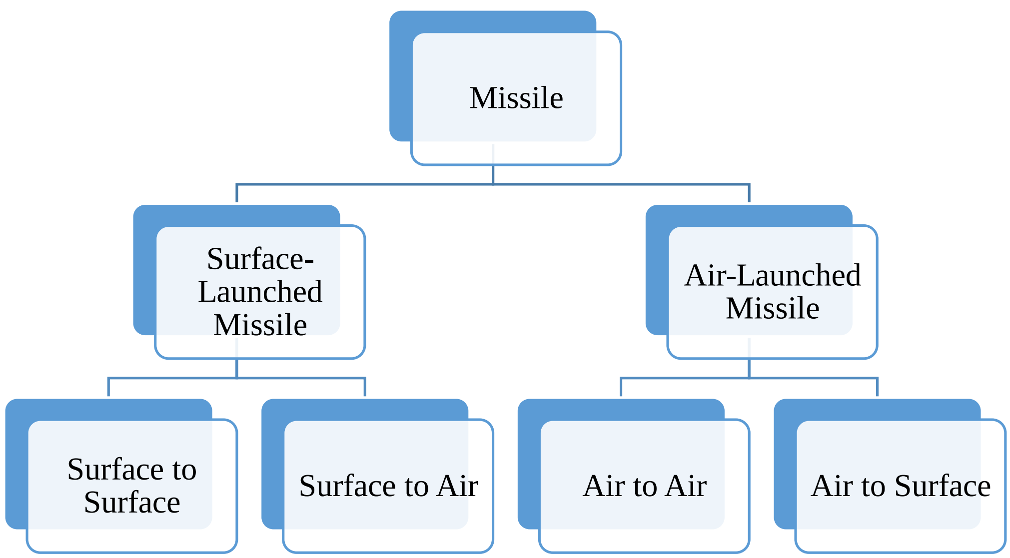


Figure 1: Missile Categories

The Air-launched missile are launched from an aircraft to hit the target in the air or ground. The major constraint for an air-launched missile is the size and weight. It small size and light weight with high thrust and Mach number is required. Therefore, the designing of combustion chamber and nozzle must be safe and efficient. The all type of missiles, whether air-launched or surface-launched, are divided in four basic sub-parts as shown in figure 2. In this thesis, we will only be discussing the propulsion system.

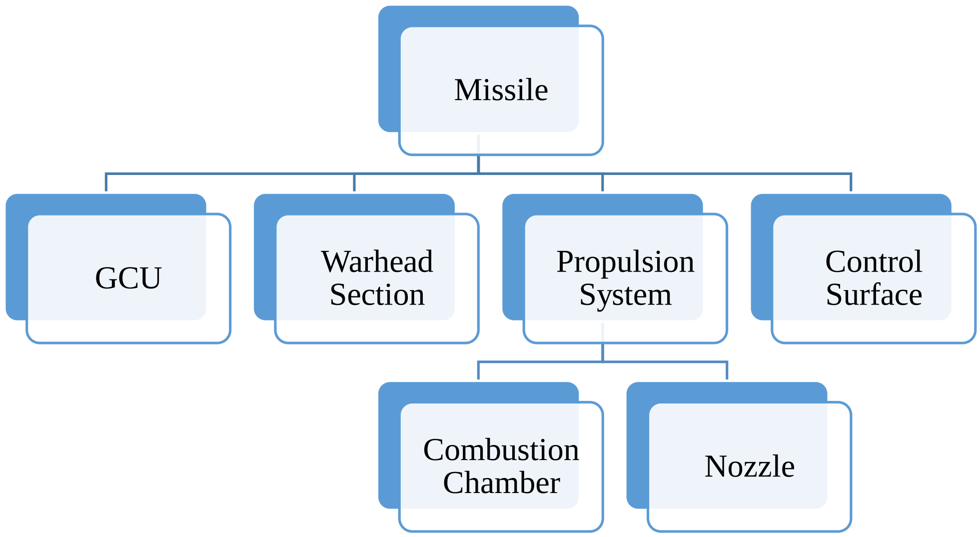


Figure 2: Missile Sub-Parts

1.1.1 Propulsion System

In Jet propulsion, high speed matter is ejected to generate enough momentum to produce motion. The burning of the fuel and oxidant is happens inside the combustion chamber, producing very high pressure and high temperature gases. Those gases ejects from the nozzle to produce motion to the missile as a reaction. The Jet propulsion in broader form is further classified into two major categories i.e. Duct propulsion and Rocket propulsion as shown in Figure 3.

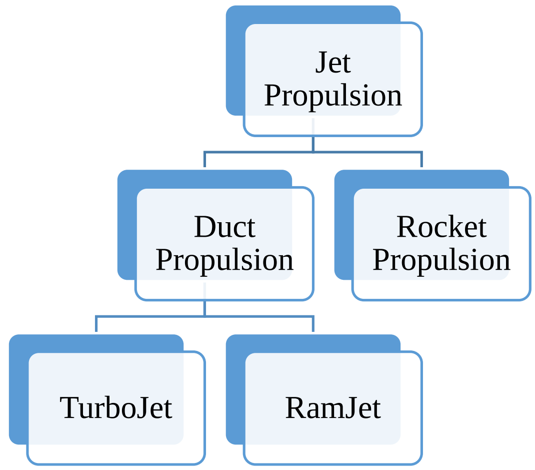


Figure 3: Jet Propulsion Hierarchy

Duct propulsion includes turbojet and ramjet engines, the common name for these engines is widely used as “air breathing engines”. The basic working principle of air breathing engines is that they take the atmospheric air and utilize it together with already stored fuel to produce thrust. On the other hand, Rocket propulsion is the class of jet propulsion in which the thrust is produce by ejecting stored matter, called and high pressure gases.

Propellants contain both the fuel and oxidant. Propellants can be in solid form or liquid. In liquid propellants, the fuel and oxidant are kept separately. On ignition, the oxidant and the fuel is fed under pressure from tanks to the combustion chamber during burning process. On the other hand, the solid propellant to be burned is contained within combustion chamber. The solid propellant charge is called “grain” (shown in Fig-4) and it contain all the elements required during burning process. The fuel and oxidant are pre-mixed homogenously with few other ingredients. Once ignited, it normally burn smoothly at predetermined burning rate from all exposed surfaces. The resulting high pressure gases flow through the supersonic nozzle to produce thrust (Fig-4) [1].

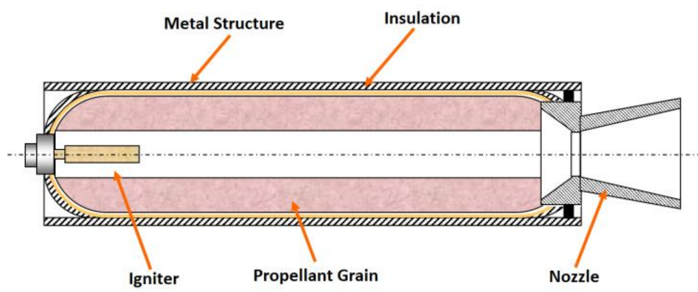


Figure 4: Solid rocket motor schematic diagram

Therefore, the nozzle is mainly used to control the velocity, direction and required parameters of the gases flow. Depending upon the variation in pressure produced in combustion chamber the nozzles are designed to operate in all flow region like subsonic, sonic, supersonic and hypersonic. The design of the supersonic nozzle remained a challenging task in fluid mechanics to achieve better performance. In a supersonic nozzle, it is not only the physical parameters of the nozzle that play an significant role, but the thermodynamic parameters of the exhaust gases flow also play a critical role in defining the design of a nozzle [2]. The converging-Diverging Nozzle known as de Laval's the most common and efficient design in rocketry. The chemical potential energy produced in the combustion chamber (rocket motor) is converted into kinetic energy using Nozzle [3]. The nozzle converts the high pressure, high temperature and low velocity (subsonic) gases into low pressure, low temperature and high velocity (supersonic) gases, hence producing high thrust [4]. To achieve desired objectives, De Laval (scientist) found that the most efficient conversion occurs when the nozzle area converges till minimum area, named as throat area, where the flow travels at sonic velocity, followed by a divergent section of the nozzle which accelerates the gases to supersonic or hypersonic velocities based on the design [5]. The exit velocity achieved in a converging-diverging nozzle is governed by the area ratios and pressure ratios [6].

1.2 Problem Statement

In converging-diverging nozzle the hot gases are transformed from subsonic to supersonic region when passes through the nozzle. In diverging section of the nozzle, the expansion of gases occurs and there is a possibility of creation of a shock-wave mainly depending upon the pressure ratio and divergence angle. The exit flow in divergent section often has strong gradients of pressure, velocity and temperature in axial and radial direction. This create non-uniformity in the flow as well. The shock inside diverging section of the nozzle and the non-uniformity of flow is undesirable. It can affect the maximum thrust produced and can cause physical damage to the nozzle.

1.3 Objective

The objective of this thesis is to design different nozzles with variation in their exit radius in order to study the behavior of shock-wave and minimize it inside converging section of the nozzle. The length of the nozzle and throat radius is considered constant in all the cases. The standard k-ε turbulence model is applied and compared with inviscid flow inside the nozzle. The performance of designed nozzle is also compared with already existing model to validate the results.

1.4 Nozzle Theory

Nozzle is integral part of rocket. It converts the potential chemical energy of the stored propellant into kinetic energy and produces thrust to lift-off the complete rocket body. The De-Laval nozzle (Converging-Diverging) consists of a convergent section which begins where the combustion chamber ends, followed by a region of a minimum area known as throat which further leads to the divergent part of the nozzle, where flow expands and expels from the exit of the nozzle.

1.4.1 Thrust

It is important to know the basic parameters and governing equations of the rocket motor nozzle before doing the computational analysis of flow inside nozzle. The equation 1 is the driving equation of the rocket nozzle. It shows the dependence of thrust on the factors like mass flow rate, exit velocity, pressure difference and exit area of the nozzle.

$$F = \dot{m}v_2 + (p_2 - p_3)A_2 \tag{1}$$

Where

$\dot{m}$ Mass flow rate

v_2 Exit / Exhaust velocity

p_2 Exit Pressure

p_3 Atmospheric Pressure

A_2 Exit Area of the Nozzle

It is pertinent to mention here that the atmospheric pressure varies with altitude, hence variation in the thrust produced. Atmospheric pressure cannot be controlled but the exit pressure and exit velocity is in our control while designing of a rocket nozzle. It is obvious from the equation 1 that the thrust will be highest when the atmospheric pressure is 'zero' i.e. vacuum conditions and it will be optimum when the exit pressure equals to the atmospheric condition [7]. Mass flow rate is another parameter on which thrust depends. Mass flow rate remains relatively constant when the nozzle has achieved the choked condition, means maximum mass flow rate is achieved at the throat. Therefore, its dependence is not variable during the flight of rocket motor.

1.4.2 Exhaust Velocity

Equation 2 shows that the exit velocity is the function of the pressure ratio, the ratio of specific heat, the temperature inside chamber, and the gas constant.

$$v_2 = \sqrt{\frac{2k\ RT_1}{k-1}\ [1 - (\frac{p_2}{p_1})^{\frac{k-1}{k}}]} \qquad (2)$$

Where

k Specific Heat ratio

R Ideal Gas Constant

T_1 Chamber Temperature

$\frac{p_2}{p_1}$ Nozzle Pressure Ratio

This shows that any increase in chamber temperature will improve the performance of the nozzle. The pressure ratio have a significant effect on the exit velocity. The lower the value of exit pressure the better will be performance until a certain value of pressure, which depends upon the atmospheric pressure as well. There is a phenomena of over and under expanded nozzles which will be described later in this chapter in detail to understand the dependence of nozzle performance on the exit pressure. In general, equation 3 shows the formula to find out pressure ratio if Mach No is known.

$$\frac{p_0}{p} = [1 + \frac{1}{2}(k-1)M^2]^{\frac{k}{k-1}} \tag{3}$$

Where

M Mach No

k Specific Heat ratio

And the Mach No is dimensionless parameter and is used to define the ratio of flow velocity to the acoustic velocity.

$$M = \frac{v}{a} = \frac{v}{\sqrt{kRT}} \tag{4}$$

Where

v Velocity of flow at any point

a Acoustic Velocity / Velocity of sound in that medium

1.4.3 Throat Parameters

We are discussing the converging-diverging nozzle in which the area decreases to minimum and increases again. The minimum nozzle area is called the throat area. The parameters like pressure, velocity, and temperature at the throat are important during design [8]. The ratio of the nozzle exit area and the throat area is called the area ratio and it is important design parameter (equation 5).

$$\varepsilon = \frac{A_2}{A_t} \tag{5}$$

$$P_t = P_1 \left\{\frac{2}{(k+1)}\right\}^{\frac{k}{k-1}} \tag{6}$$

$$T_t = \frac{2T_1}{(k+1)} \tag{7}$$

$$v_t = \sqrt{\frac{2k}{k+1} RT_1} \tag{8}$$

Equation 6 shows the pressure at the throat, the pressure at which the mass flow rate is maximum is called the critical pressure, above which is chocking condition is established. It is important to note that the chocking condition only occurs at the throat. Similarly, equation 7 and equation 8 shows the temperature and velocity at the throat respectively. In an ideal converging-diverging nozzle the velocity at the throat is sonic, i.e., Mach no is 1.

1.4.4 Under- and Over-Expanded Nozzle

Under-Expended nozzle is the nozzle in which the expansion of gases doesn't completely happen inside the nozzle divergence area, therefore, the exit pressure is higher than the atmospheric pressure. Optimum expansion is not achieved at the exit, rather expansion of gases occur outside the nozzle [9].

On the other hand, Over-expanded nozzle is the nozzle in which the exit pressure is lower than the atmospheric pressure. The area of the nozzle is too large for optimum expansion at the exit and expansion of the gases occurs inside the nozzle. There are chances of producing shock wave in this case [10].

There is only one altitude where the exit pressure becomes equal to the atmospheric pressure and the flow expands optimally at the exit.

1.4.5 Design Parameters

There are few design parameters of the nozzle and based on those parameters the performance of the nozzle is calculated. The design parameter includes pressure inside chamber, the chamber temperature, area ratio, and divergence angle of the nozzle. As described earlier the performance of the nozzle is mainly calculated by the thrust produced or the specific impulse. The Specific Impulse is used to measure the efficiency of the rocket motor, it is the thrust produced per unit mass of the propellant. Therefore, the pressure ratio, exit velocity and the thrust produced are mainly the performance parameters.

Chapter 2

Literature Review

As discussed in Chapter 1 that there are many design parameters of the nozzle which are inter-related to each other. On the other hand we require optimum performance of the nozzle for maximum time. It is advice-able to change one design parameter at once and observe its effect on the performance keeping other parameters constant in accordance with the design. There were many studies carried out to design the nozzle and analyze the fluid flow inside it using CFD.

Natta P. et.al [11] worked on a topic "Flow analysis of rocket nozzle using computational fluid dynamics (CFD)" and they kept the throat diameter, inlet diameter and Mach number constant at Mach 3 and varied the divergence angle in their nozzle design. They observed the effect of variation in divergence angle on various parameters like velocity, temperature, and pressure. They concluded that the velocity magnitude at the throat is same for all the deviation degrees of angle and it is calculated as 260 m/s. Another conclusion was that we can go with only 30 to 40 degrees of divergent angle conical nozzle for the maximum velocity.

Bogdan et.al [12] researched on the topic "Analysis of Flow in Convergent-Divergent Rocket Engine Nozzle Using Computational Fluid Dynamics" and studied the proper geometrical design of the nozzle and how regulating the gases in the nozzle will affect the velocity. They used a finite volume rewarding code to know about the flow through convergent deviating nozzle. After completing their simulation of the design successfully, they observed that the nozzle created on the base of exit parameters is in concurrence with the scope.

Pandey et.al [13] studied the topic "CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software" and they analysed the effect of variation in divergence angle on the performance and behavior of nozzle keeping the Mach number constant at 3. 2-D modelling was carried out. The nozzle was designed using method of characteristics and the supersonic flow properties were investigated in this study. The effect of viscosity and turbulence were observed near the walls. They concluded that the efficiency of nozzle is highest between the 12 and 16 degree angles.

Ramji V et.al [14] worked on the matter “Design and Numerical Simulation of Convergent Divergent Nozzle” and studied about the designing of C-D nozzles to understand the flow at supersonic or hypersonic speeds. They have used method of characteristic for designing a nozzle. The variable parameter was the Mach number and its effect on the minimum length in the diverging section was observed. Hence, the variation in exit area was also observed with the change in divergence length. The physical parameters of the throat were kept constant in all the designs. The Ansys Fluent was used to carry out the CFD of the designs. Two turbulence models the standard k-ε and spalart Allmaras were used and results were compared. They observed that the values of exit area obtained from the CFD were found close to the theoretical exit area.

Sher Afghan Khan et.al [15] studied the topic “CFD analysis of cd nozzle and effect of nozzle pressure ratio on pressure and velocity for suddenly expanded flows” and modelled a converging-diverging nozzle. The aim of the study was to investigate the effect of NPR on the exit velocity. The sudden and rapid expansion of the flow is the main problem in supersonic regions. This effect was also incorporated and analyzed in this study. For detailed results and further analysis, the standard k-ε turbulence model with standard wall friction was used in ANSYS Fluent. The suddenly expansion duct has the L/D ratio 10. The results showed that the velocity has increased as the pressure decreased and it was according to the theoretical calculations.

Md Touseef Ahmad et.al [16] conducted a research on the topic “Modeling and simulation of Convergent-Divergent Nozzle Using Computational Fluid Dynamics” to study the very basic concept of designing a nozzle using empirical formulas and the doing the CFD in ANSYS Fluent to validate the results. The variable parameter here was the divergence angle, while other parameters were kept constant. The parameters like pressure, temperature and velocity were studied at the post processing of CFD results. The results which were obtained from post processing were also compared with the analytical results calculated by using basic nozzle equation. The comparison showed close resemblance between both the results.

Ch. VARUN et.al [17] studied on the topic "CFD Analysis of Convergent-Divergent Nozzle" and studied the analysis of flow within Convergent-Divergent supersonic nozzle of different cross section shapes and areas. The types they have used are the circular, rectangular and square. The aim of the study was to investigate the effect of these cross sections on the exit velocity. The parameters at the output like pressure, temperature and velocity were analyzed in all the cross section. The input boundary conditions were kept similar for all the cases. They have used CATIA for the designing and ANSYS Fluent for the CFD analysis. The results showed that the performance of rectangular cross section was better than the other two.

Dr. V. Chitti et.al [18] studied a topic "CFD Analysis of Convergent- Divergent Supersonic Nozzle" and they analysed the effect of variation in divergence angle on the performance and behavior of nozzle keeping the Mach number constant at 4. 2-D modelling was carried out. The nozzle was designed using method of characteristics and the supersonic flow properties were investigated in this study. The effect of viscosity and turbulence were observed near the walls. They concluded that the efficiency of nozzle is highest between the 13 and 15 degree angles. Moreover, the computational results were in a good agreement with the theoretical results.

Mohan Kumar et.al [19] researched on the topic "Design and Optimization of De Lavel Nozzle to Prevent Shock Induced Flow Separation" and they designed a Converging-Diverging nozzle (De Laval) using RAO's parabolic method. The design was focused to optimize the performance of the nozzle by achieving exit pressure close to the atmospheric pressure. The shock can induce inside the nozzle as well, in over-expanded nozzles, this phenomena was also prevented to enhance the performance of the nozzle. The results were analyzed and they concluded that the optimum expansion is very critical for efficient performance of the nozzle. This was achieved without any adverse (Shock or flow separation) pressure gradient.

A.Shanthi Swaroopini et.al [20] worked on a topic "Numerical Simulation and Optimization of High Performance Supersonic Nozzle at Different Conical Angles" and studied the effect of variation in divergence angles on the performance parameters like the nozzle pressure ratios, area ratios and the thrust produced. The results obtained were compared with the theoretical results as well. The design parameters other than the divergence angle were kept constant for all the design and input boundary conditions were same for all the designs. The performance of the nozzle having divergence angle of 11 degrees was the best amongst all the designs.

We can summarize that a study was carried out by keeping the Mach number constant and varying the divergence angle to observe its effect on various parameters like velocity, temperature, and pressure. Another study was carried out to design a nozzle by varying the exit Mach number and observe its effect on the length of the nozzle, variation in pressure and velocity while keeping the throat area constant in all the cases. Further, a study was carried out where the divergence angles were varied and observed its effect on Mach number, pressure and exit velocity. It showed that the divergence angle up to specific limit provides better results. In supersonic nozzles, the sudden expansion of gases can produce shock inside nozzle due to flow separation. A good nozzle design must be optimized to achieve maximum thrust without producing shock due to flow separation. Hence, various simulations have been carried out to choose the best nozzle design for maximum thrust based on the same input conditions.

Chapter 3

CFD Modeling and Simulation Setup

This chapter describes the brief background of flow solver and basic governing equation being used. The tool which is used to carry out our simulations and the turbulence model being used is also covered in this chapter. At the end, the nozzle dimensions and its setup in software is described. ANSYS Fluent was used for all type numerical results. ANSYS design modeler and mesh generation tools were used for design and grid generation of the nozzle respectively.

Computer simulation of nozzle was done using computational fluid dynamics. Computational Fluid Dynamics (CFD) is an engineering tool commonly used to simulate the complex models using physics and mathematical equations numerically. Its analysis provides detail information on all useful parameters like velocity, pressure, temperature, density, turbulence, and multiphase flow which assists the experimentation. We can predict the expected results under different conditions of a problem using simulations before practical testing. Nozzle design and modeling is a complex task involving various parameters and equations as mentioned above. The CFD helps not only in the modeling of the nozzle; also the model can be optimized by analyzing the results under different atmospheric and boundary conditions. Thus, it will make easier to predict the real-time behavior of a nozzle under various conditions before experimentation.

3.1 Basic Equations

The primary governing equations are the equation of conservation of mass or continuity equation and equation of conservation of momentum.

3.1.1 Continuity equation or mass conservation equation in the differential form is

$$\frac{\partial \rho}{\partial t} + \nabla . (\rho \vec{v}) = S_m \qquad (9)$$

3.1.2 Conservation of momentum as

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla . (\rho \vec{v} \vec{v}) = -\nabla p + \nabla . (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \qquad (10)$$

3.2 Standard k-ε (SKE) Model

The standard k-ε turbulence model in ANSYS Fluent has become the workhorse of practical engineering flow calculations [21, 22]. The standard k-ε model (SKE) is a model which is based on two equations, the model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). Moreover, to simplify the things, in this model, it is further assumed that the flow is turbulent and other fluid effects like molecular viscosity are negligible. Therefore, it is only valid for turbulent flows.

3.2.1 Transport equations for the standard k-ε model

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\frac{\mu+\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \quad (11)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \quad (12)$$

3.2.2 Modeling the turbulent viscosity

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \quad (13)$$

Whereas, C1ε, C2ε, Cμ, σ_k and σε are the constants in the abovementioned equations and their default values are mentioned below [23].

C1ε = 1.44, C2ε = 1.92, Cμ = 0.09, σk = 1.0, σε = 1.3

Computational Fluid Dynamics (CFD) is extensively used for analyzing fluid flow by solving governing equations of fluid dynamics. CFD can be used to solve and analyze complex numerical problems involving multiphase flow and interaction. It has great potential to simulate for a better design and optimize it rather than wasting time and cost on prototypes [24]. In Engineering, CFD is widely used and it gives better understanding and visualization to the problem. It is involved with physical laws in the form of partial differential equations called as Navier-Stroke equations [25]. The CFD analysis consists of the following steps.

- Modeling
- Meshing
- Pre-Processing
- Solver

- Post-Processing

Here, the modeling and meshing is carried out in Design Modeler and ANSYS Mesh. ANSYS Fluent is used as solver tool.

3.3 Nozzle Modeling and Meshing

It is described in Chapter 1 that there are few design parameters which directly affect the performance of the nozzle. Area Ratio (ε) is one of those parameters. In this thesis we have designed and studied 06 different models to investigate the behavior of flow inside the nozzle. The overall length and the throat area is kept constant in all designs. However, the exit radius is variable parameter. The proposed sketch of the nozzle is shown in Figure 5. The total length is 0.6m in all the cases with variable diameter along the axis making it converging-diverging nozzle.

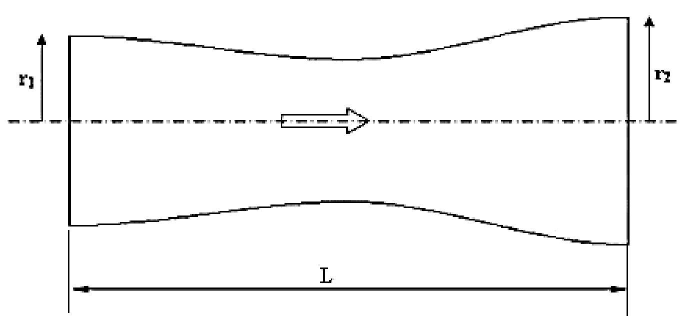


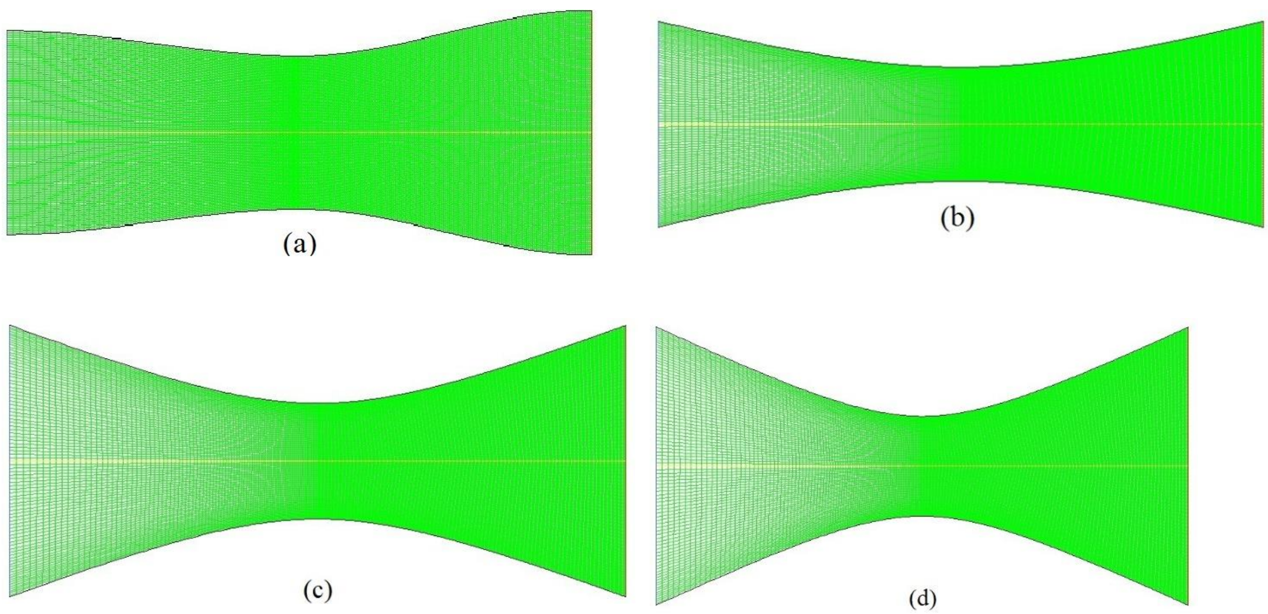
Figure 5: Sketch of the proposed nozzle

The designing, modeling and mesh generation the Convergent-Divergent Nozzle was carried out in Design Modeler and ANSYS Mesh. The basic dimensions of the all the designs are:

Table 1. Nozzle Dimensions

Designs	Length of the Nozzle (m)	Nozzle Exit radius (m)	Nozzle Throat Radius (m)	Area Ratio (ε)
(a)	0.6	0.12	0.075	2.56
(b)	0.6	0.10	0.06	2.78
(c)	0.6	0.13	0.06	4.69
(d)	0.6	0.15	0.06	6.25
(e)	0.6	0.17	0.06	8.03
(f)	0.6	0.19	0.06	10.03

The Design Modeler application provides persistent and parametric feature-based modeling environment that is a perfect tool for design optimization. Its modeling standard is to outline a 2D sketched profiles and use them to generate features. The advantage of Design Modeler over other modeling tools is that it is easy to use, user-friendly interface and it has a function to divide a model into small separate surfaces during meshing for improved mesh generation. As design modeler has the provision of producing finer mesh at the desired area. Therefore the mesh grid adjacent to the wall is finer as compared to the center region. The purpose is for the fine mesh is to capture the small gradients near the walls efficiently [26]. The meshing of all the designs was carried out and displayed in Figure 6.



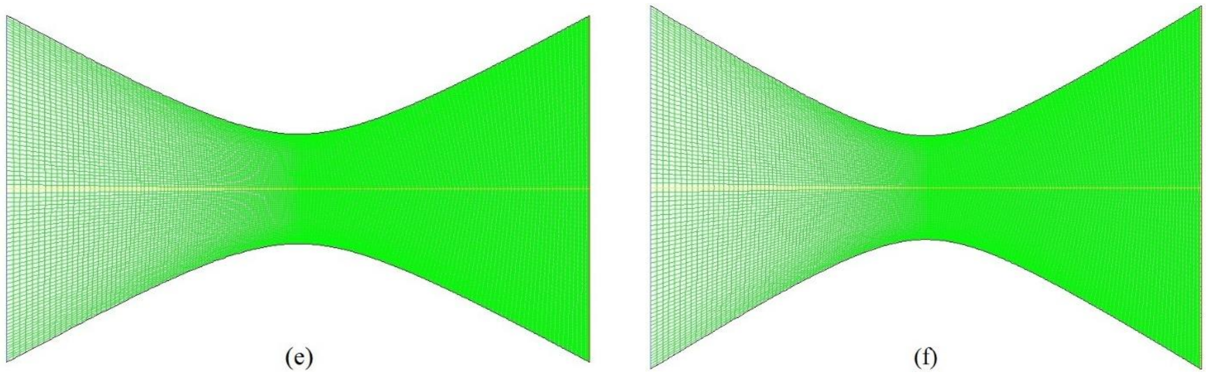


Figure 6: Meshing of All models

3.4 Solver Setup

ANSYS Fluent is used for computational analysis of the fluid flow inside the nozzle. This analysis includes various performance factors like inlet pressure, inlet temperature, inlet velocity / Mach number, outlet pressure outlet temperature and outlet velocity / Mach number [27]. The solver procedure is shown in figure 7.

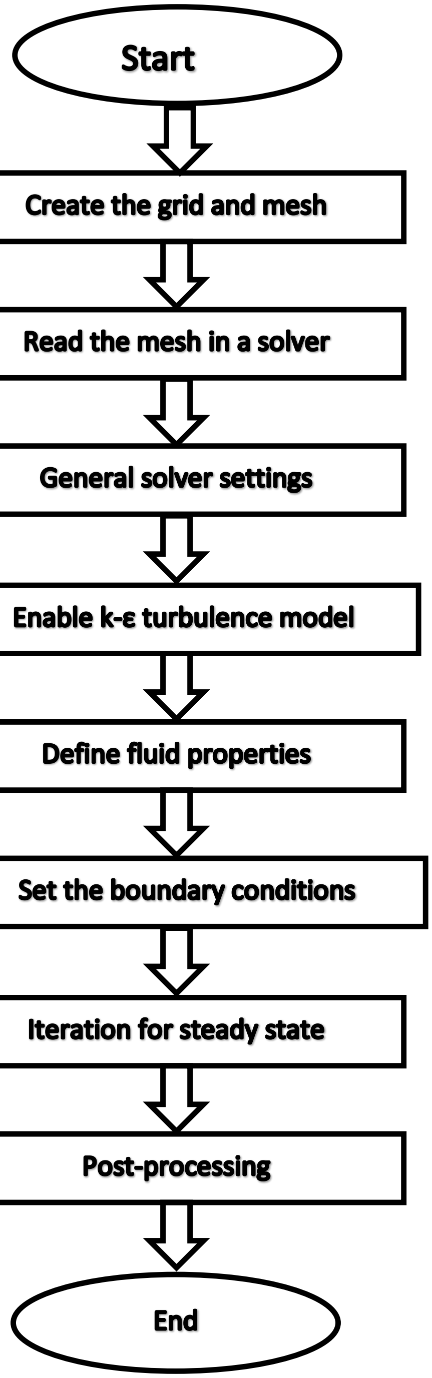


Figure 7: Simulation procedure for CFD model

Pre-Processing involves the transformation of physical problem statements into the software. The type of material, fluid, and boundary conditions are defined as shown in table 2. The boundary conditions were given according to a standard available in the literature to validate any small scale nozzle design [28]. The desired energy and flow models are selected and assumptions are made concerning the nature of the flow, whether it's viscous or inviscid, compressible or non-compressible, steady or non-steady [29].

Table 2. General Setup Information:

Setup	
Solver Type	Pressure-Based
2D Space	Axisymmetric
Time	Steady
Boundary Conditions for Case 1	
Pressure at Inlet	400 KPa
Temperature at Inlet	300K
Pressure at Outlet	100 KPa
Temperature at Outlet	300K
Boundary Conditions for Case 2	
Pressure at Inlet	240 KPa
Temperature at Inlet	300K
Pressure at Outlet	100 KPa
Temperature at Outlet	300K
Boundary Conditions for Case 3 (Inviscid Flow)	
Pressure at Inlet	240 Kpa
Temperature at Inlet	300K
Pressure at Outlet	100 KPa
Temperature at Outlet	300K

The abovementioned conditions were given to all the designs in ANSYS Fluent to analyse the flow and the performance parameters. The design with better results was selected and then it was compared with already available design in literature with the same boundary conditions to validate our results.

Chapter 4

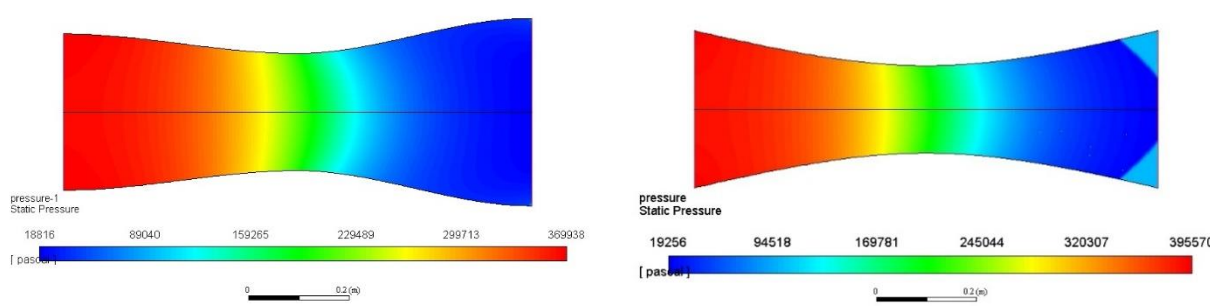
Results and Discussion

The nozzle plays a vital role in producing thrust and controlling the speed and direction in different application like aerospace and rocketry. In this study, simulations were performed to carry out an analysis of fluid flow inside nozzle, shock simulation inside nozzle using the k-ε model and then comparing it with the inviscid flow. The simulation results and analysis of each case is discussed below.

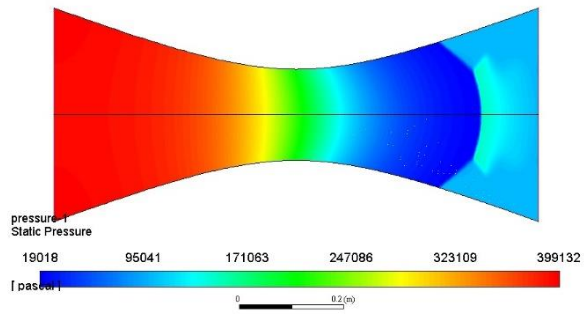
4.1 Case I: Fluid Flow in Converging-Diverging Nozzle (SKE Model)

This case was established to study and analyse the full flow inside the rocket nozzle. Therefore, the pressure at the inlet was set to 400KPa. The pressure at the outlet was kept close to the atmospheric pressure. However, the temperature at the inlet and outlet was kept at 300K. The standard k-ε turbulence (SKE) model was used to analyze the fluid flow inside the nozzle and the behavior of flow at the exit. These conditions and turbulence model was applied to all designs. The behavior of pressure, temperature and Mach number contours is discussed below. The velocity vectors along the axis of nozzle is also shown and discussed for all the designs.

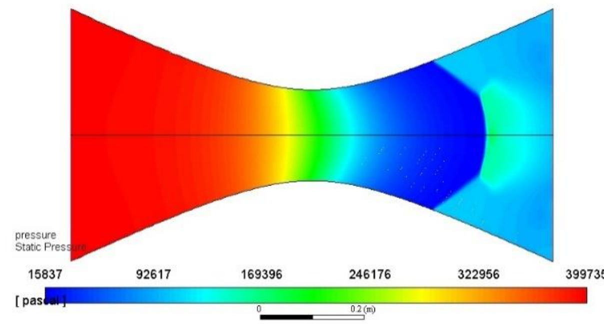
4.1.1 Pressure Contours



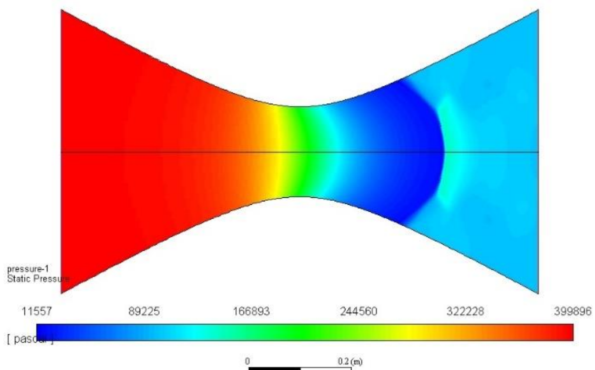
(a) **(b)**



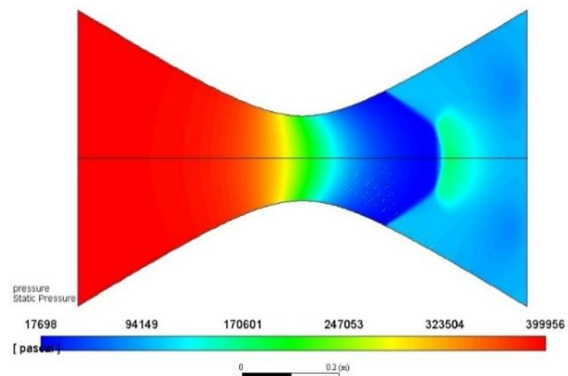
(c)



(d)



(e)



(f)

Figure 8: Pressure Contours

The Figure 8 displays the pressure contours of all the designs for Case I. The red color showing the highest value of pressure and blue showing the lowest. The Pressure at the inlet is 367Kpa in design (a) however, it is 399KPa in all other designs. The flow separation can been seen clearly at the exit of the nozzle in design (b) [30]. The divergence is relatively high in design (c, d, e, and f), therefore there is a sudden expansion of flow is observed and pressure dropped rapidly, producing a shock inside the nozzle. There is neither a flow separation nor a shock is observed in the design (a). The pressure at the exit is observed as 18 KPa in design (a) and 19 Kpa in design (b). The exit pressure in the other designs is of no use as the shock is produced.

4.1.2 Temperature Contours

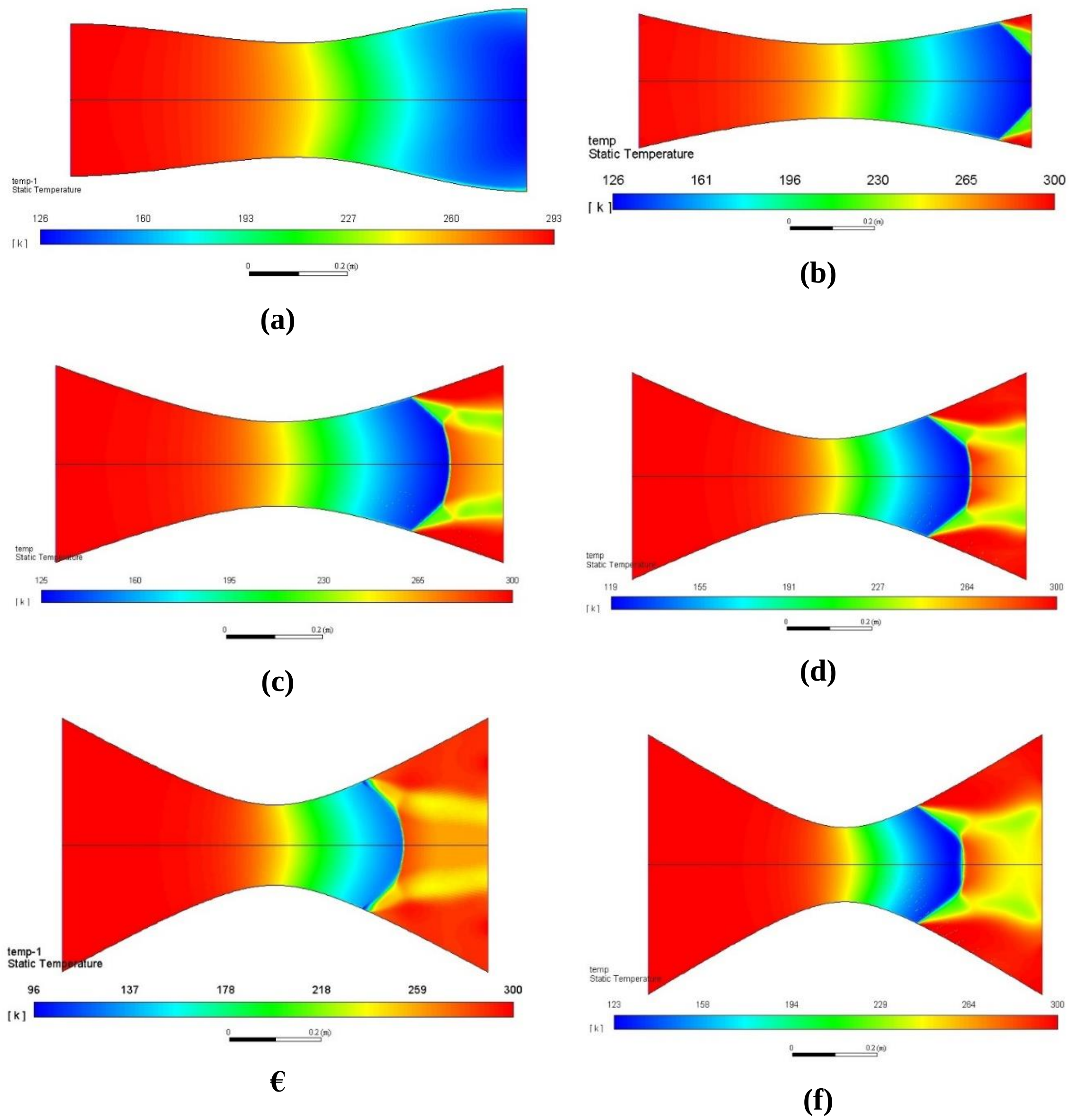


Figure 9: Temperature Contours

The temperature inside the rocket motor chamber is generally around 3000K, but in this case we are studying the exit flow properties under the SKE model and then exit gases flow properties of inviscid flow. Therefore, the temperature at the inlet is kept at 300K which is reduced during the expansion of gases in the divergent section of a nozzle to approx. 125K in all the designs as shown in **Figure 9.**

4.1.3 Mach No Contours

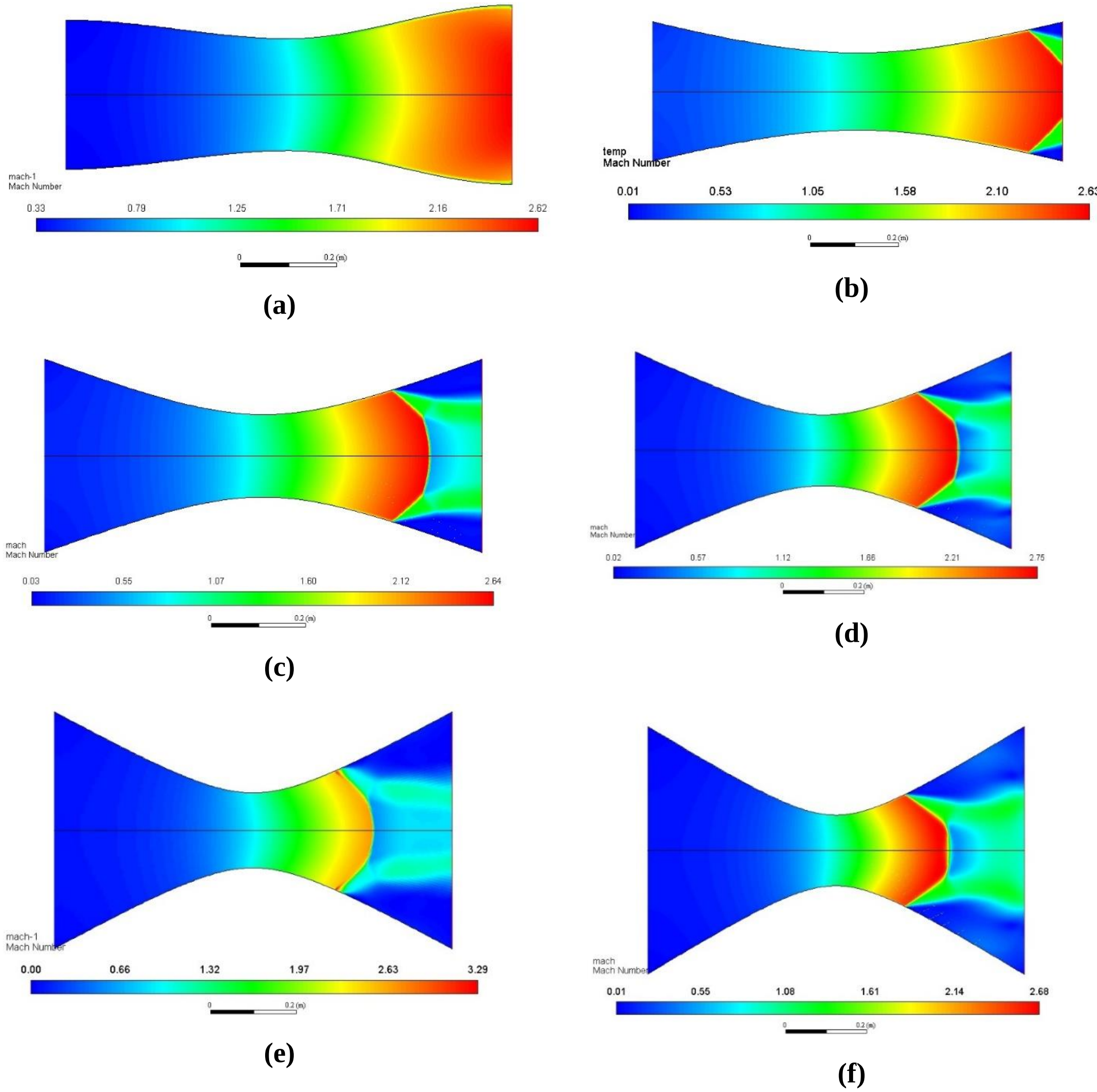


Figure 10: Mach No Contours

The Mach number is a key parameter used for calculation of thrust of the nozzle as well as it segregates whether a nozzle is subsonic, sonic or supersonic. The Mach number is close to 1 at the throat of the nozzle of all designs and the maximum achieved at the exit of design (a) is 2.64 as shown in Figure 10. Although the value of Mach number in all designs is maximum for design (d), which is 2.75, but there is a shock inside the nozzle. Similarly, the value of Mach number in design (e) reaches up to 3.29 at the edges of the shock.

4.1.4 Velocity Vectors

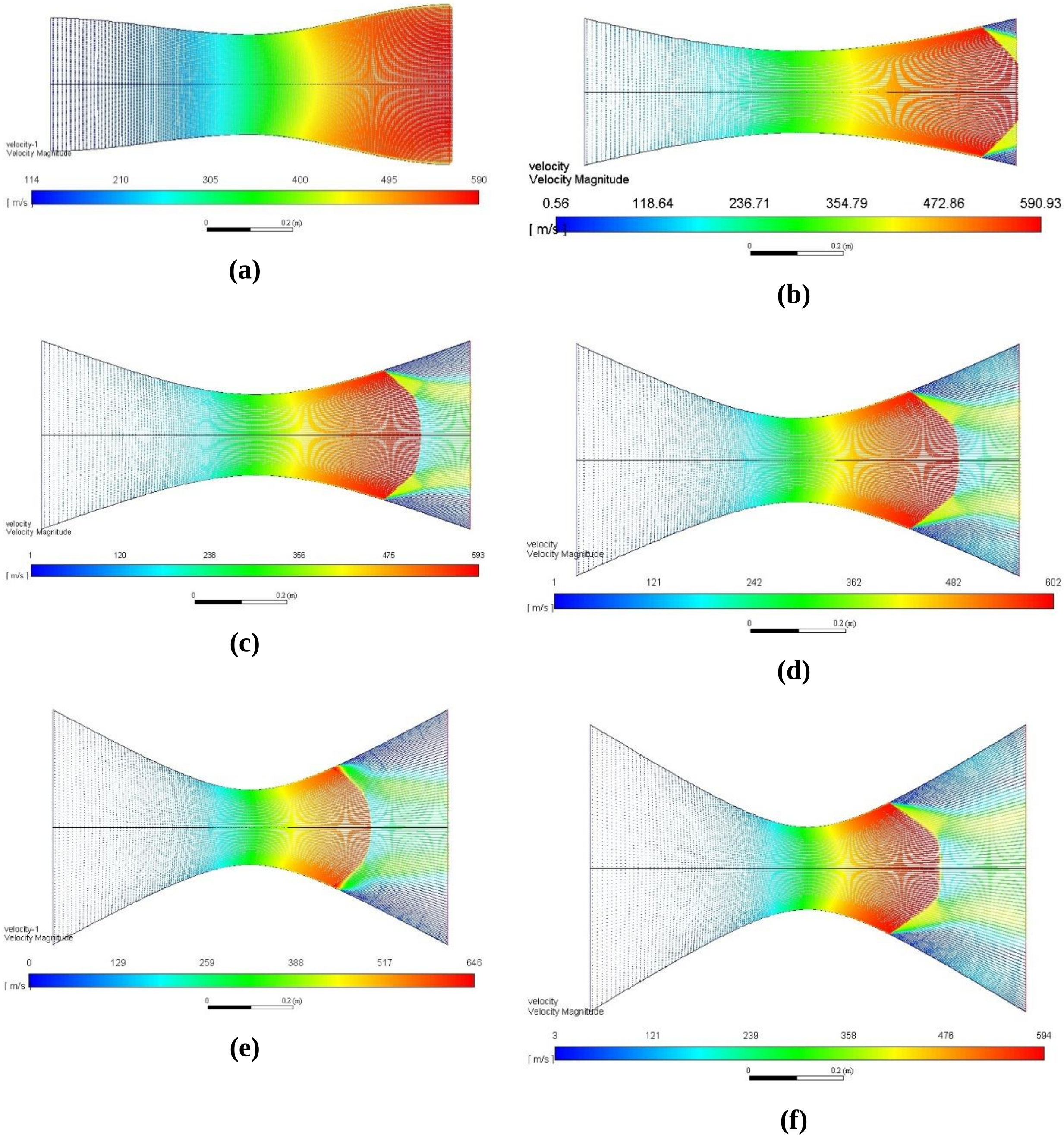


Figure 11: Velocity Vectors

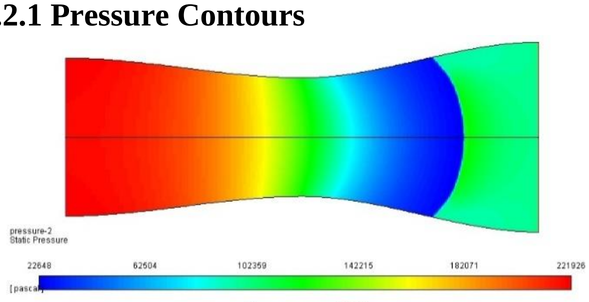
In Figure 11, the velocity of flow at the inlet is low as the pressure of the gases is higher in all the designs. As the pressure decreases, the velocity increases along the axis of the nozzle and attain its highest value at the exit in design (a) and (b), but it attain the highest value at the shock front in design (c, d, e and f). The flow separation and small vortex can be seen at the exit in design (b). The maximum value achieved in design (a) is 590 m/s.

4.2 Case II: Shockwave of Converging-Diverging Nozzle (SKE Model)

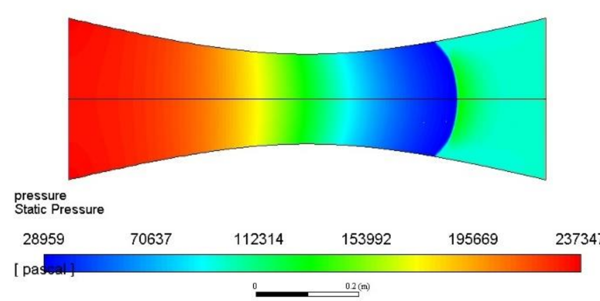
There is phenomena called over-expanded nozzle, in which the exhaust pressure of the product gases falls below the atmospheric pressure and expansion of the gases occurs very rapidly inside the nozzle, hence producing a discontinuity in the flow. This sudden discontinuity is called a shock wave. This type of shockwave generally occurs where the area ratio gradient is higher and pressure drops rapidly, often close to the exhaust end of the supersonic nozzle. The speed of the flow just before the shock has a significant impact on the shock. The higher the Mach number just before the shock, more significant would be the effect of shock on the nozzle. This can cause a reduction in thrust and sometimes physical damage to the nozzle [31].

This case was established to study abovementioned over-expanded phenomena and shock inside the nozzle. The pressure at the inlet was set to 240KPa. The pressure at the outlet was kept close to the atmospheric pressure i.e 100 KPa. However, the temperature at the inlet and outlet was kept at 300K. Same as in the case I, The standard k-ε turbulence model was used to analyze the type and the shape shock inside the nozzle and the behavior of flow at the exit. These boundary conditions and turbulence model was applied to all designs. The behavior of pressure, temperature and Mach number contours is discussed below. The velocity vectors along the axis of nozzle are also shown and discussed for all the designs.

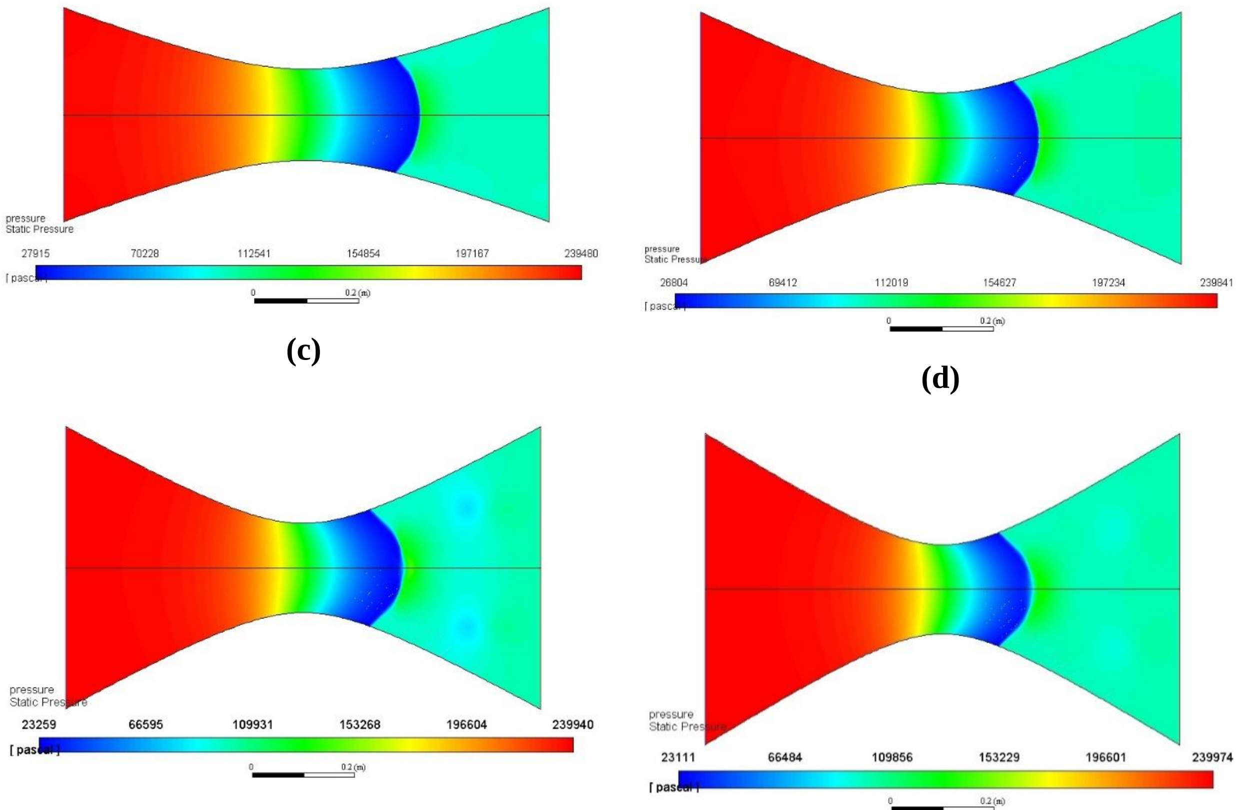
4.2.1 Pressure Contours



(a)



(b)



(c) **(d)**

(e) **(f)**

Figure 12: Pressure Contours

The Figure 12 displays the pressure contours of all the designs for Case II. The red color showing the highest value of pressure and blue showing the lowest. The Pressure at the inlet is 222Kpa in design (a) however, it is 239KPa in all other designs. In this case, the inlet pressure is very low as compared to case I. Therefore, as the flow passes through the throat of the nozzle, unlike case I, a sudden expansion happens in the divergence section of the nozzle. This produces the shock inside the nozzle diverging section. The pressure measured at the shock front was 22Kpa in design (a) much lesser than the atmospheric pressure. The pressure in design (b) was measured as 28Kpa at the shock inside the nozzle. As we know that the pressure drops more rapidly as we increase the divergence angle. This statement proved valid in our study. We can see in design (b) onwards that the pressure is reducing from 28KPa to 23KPa as the exit area is increasing from 2.56 to 10.03. The shape of the shock is curved in design (a) but as the divergence angle has increased the shape changed from curve to lambda because the loss in axial flow is higher close to wall than the axis.

4.2.2 Temperature Contours

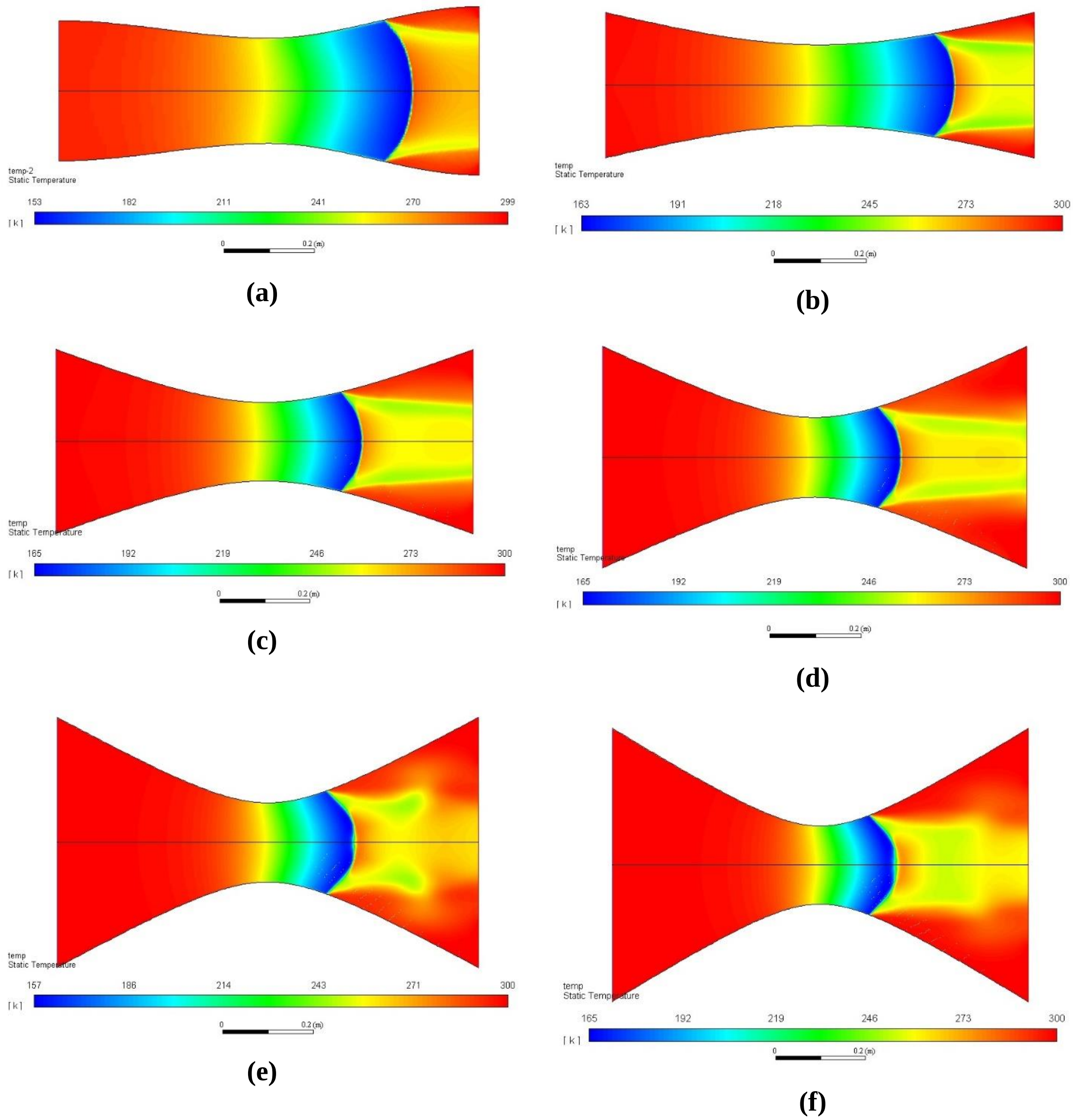


Figure 13: Temperature Contours

As discussed in case I, The temperature inside the rocket motor chamber is generally around 3000K, but we are studying the flow properties under the k-ε turbulence model. Therefore, the temperature at the inlet is kept at 300K which is reduced suddenly to 153~165 K in all the designs during the sudden expansion of gases in the divergent section of a nozzle as shown in **Figure 13.**

4.2.3 Mach No Contours

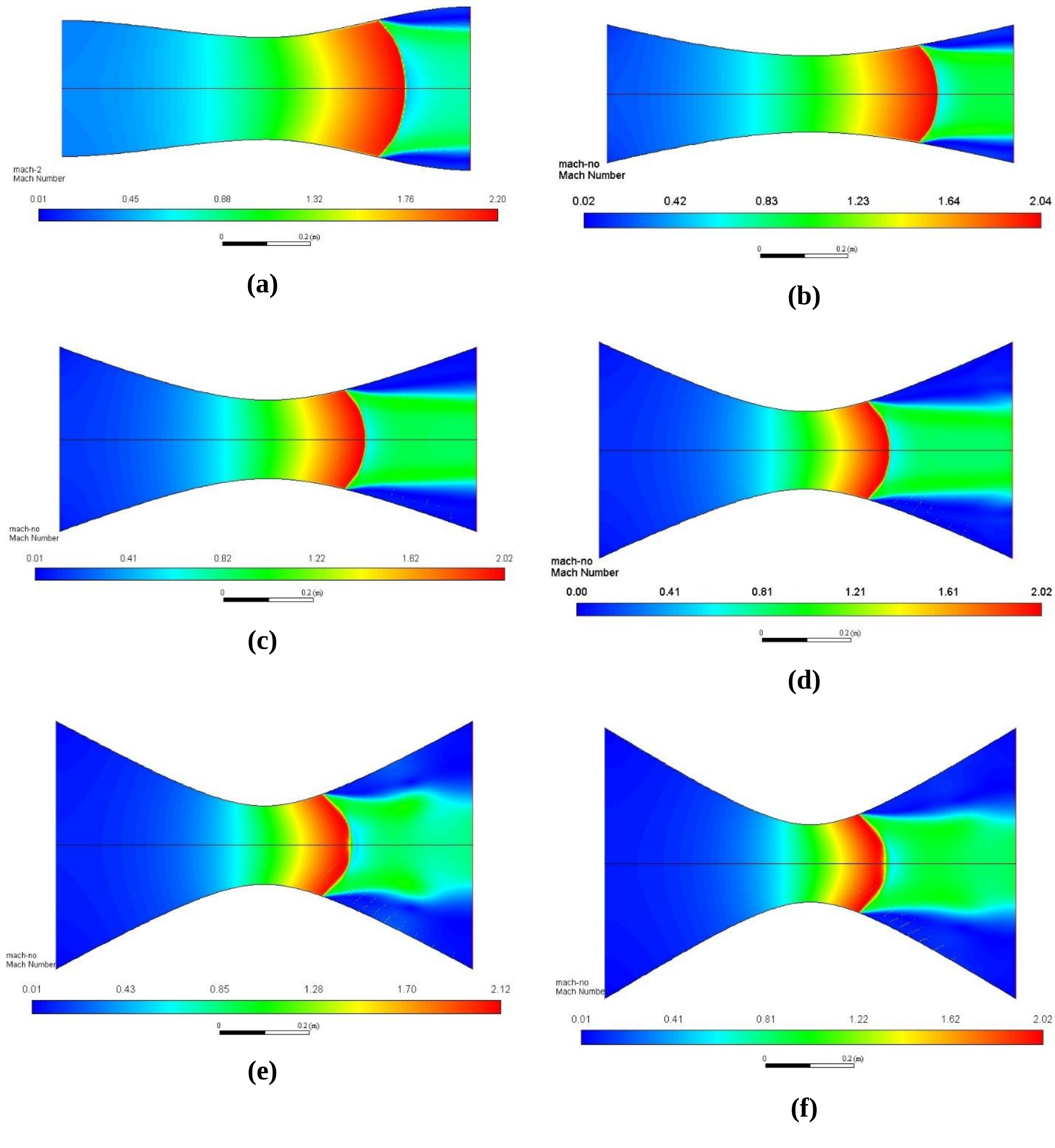
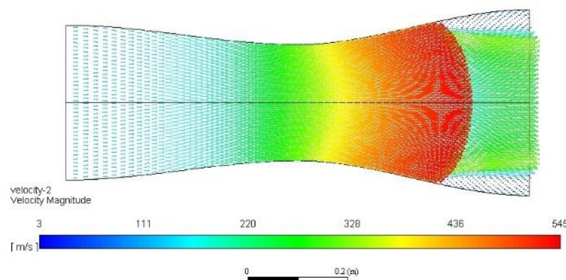


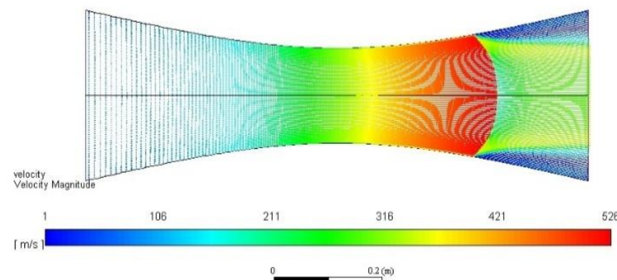
Figure 14: Mach No Contours

In figure 14, it is shown that the Mach number is close to 1 at the throat of the nozzle of all designs and it is highest at the shock front. The walls frictions are higher close to the wall of the nozzle and lesser at the center. Hence, momentum loss is higher close to the wall causing loss in velocity of the flow, thus producing shock earlier close to the wall than it takes place in the main flow. This makes the shock shape as curved in the designs with low divergence and Lambda shape shock in the designs having relatively high divergence. Mach no at the shock front is highest for the design having less divergence and it is reducing with increase in divergence. Maximum Mach No achieved at the shock front of design (a) is 2.20. It remained close to Mach 2 for all other designs.

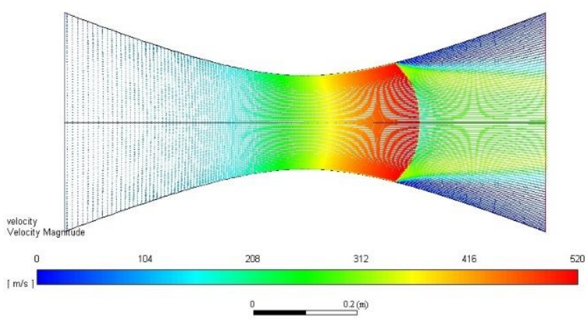
4.2.4 Velocity Vectors



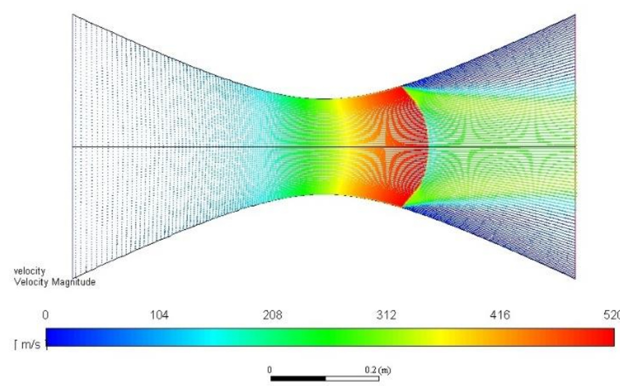
(a)



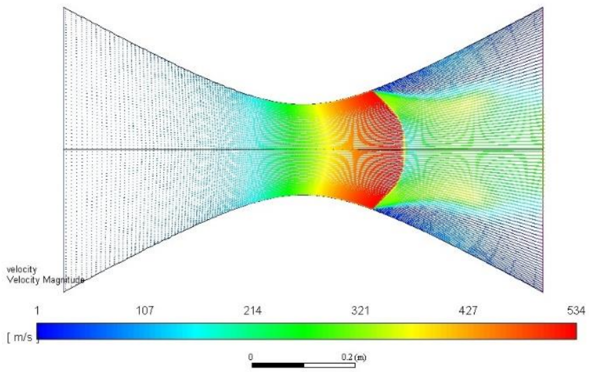
(b)



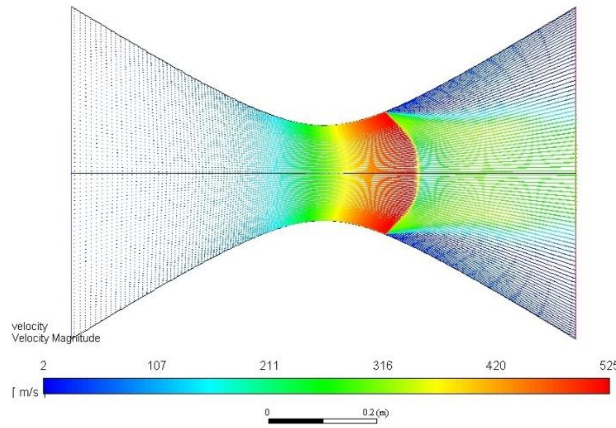
(c)



(d)



(e)



(f)

Figure 15: Velocity Vectors

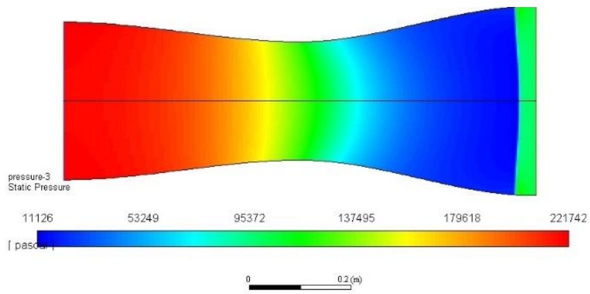
In Figure 15, Similar to Case I, the velocity of flow at the inlet is low as the pressure of the gases is higher in all the designs. As the pressure decreases, the velocity increases along the axis of the nozzle and attain its highest value at the shock. The velocity vectors of flow showing a rapid increase in the velocity of the divergent section of the nozzle. It is also showing the formation of vortex close to wall of the nozzle just after the shock because the flow separation has occurred close to the outlet of the nozzle and back pressure is generated. The vortexes are more evident and prominent in the designs with higher divergence area. The maximum value achieved at the shock of design (a) is 545 m/s.

4.3 Case III: Shockwave of Converging-Diverging Nozzle (Inviscid Flow)

Inviscid flow analysis assumes flow to be laminar and disregards the effect of viscosity on the flow. The inviscid theory predicts a simple shock structure consisting of a normal shock followed by a smooth recovery to exit pressure in the divergent part. But in reality, viscous effects of the fluid like boundary wall layer flowed by flow separation drastically alter the flow in a converging-diverging nozzle [32].

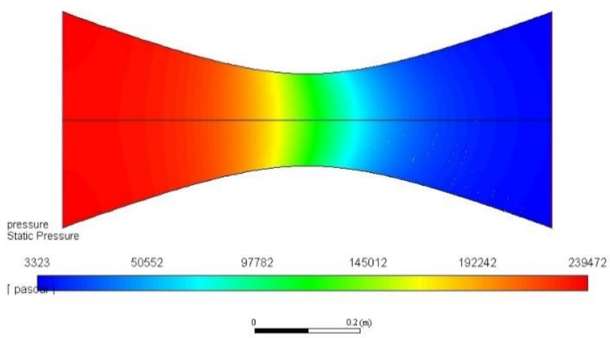
The flow velocity is observed very high in this simulation as compared to pervious cases. Therefore, we can assume the flow to be inviscid so that we can have a comparison of shock and flow characteristics with the viscous flow simulations. The inlet pressure was set to 240KPa. The pressure at the outlet was kept close to the atmospheric pressure i.e 100 KPa. However, the temperature at the inlet and outlet was kept at 300K. Instead of SKE turbulence model, the inviscid flow properties was used in ANSYS Fluent to analyze the type and the shape shock forming inside the nozzle and the behavior of flow at the exit. The behavior of various parameters like pressure, temperature and Mach number contours is discussed below. The velocity vectors along the axis of nozzle are also shown and discussed for all the designs.

4.3.1 Pressure Contours



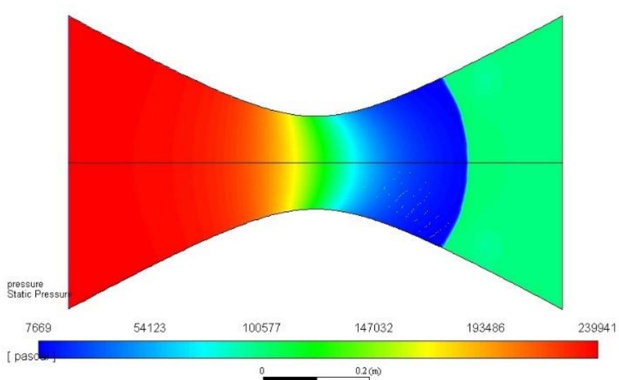
Not Available

(a)

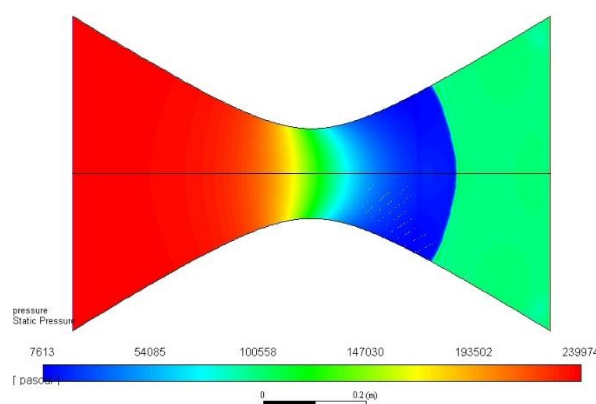


Not Available

(c)



(e)

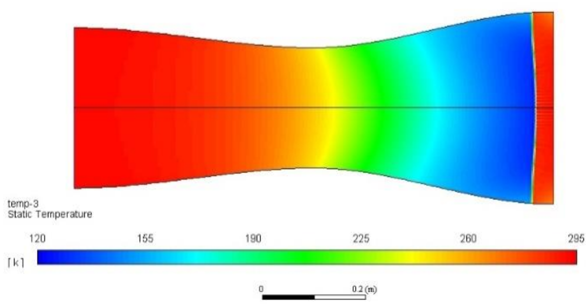


(f)

Figure 16: Pressure Contours

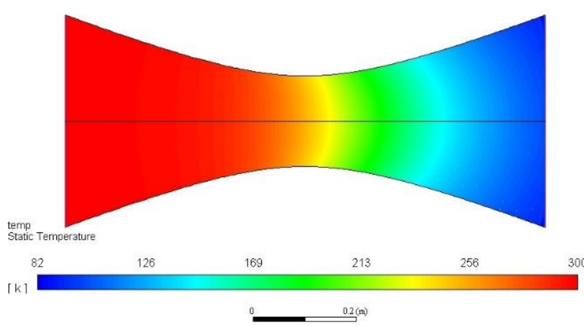
The third case is simulating the shock inside the nozzle in an inviscid flow. The Pressure at the inlet is 222Kpa in design (a) however, it is 239KPa in all other designs. The pressure contours for design (b) and (d) are not available as they were not compatible with these boundary condition. In this case, the pressure decreases in the divergent section more steadily as compared to Case II. The wall frictions and viscosity of the flow are assumed to be zero. Therefore, the shape of shock is straight in the design (a) and it changed to curved shape as divergence area is increased. In this case, the shock location is shifted close to the nozzle exit as compared to Case II, shown in Figure 16. The reason for shape of the shock to be straighter and shifting of shock close to exit is the model we have used to simulate our design.

4.3.2 Temperature Contours



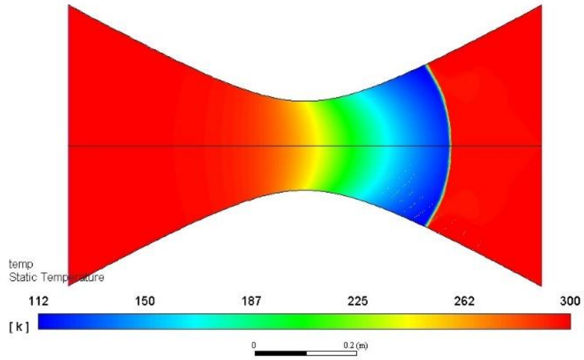
(a)

Not Available

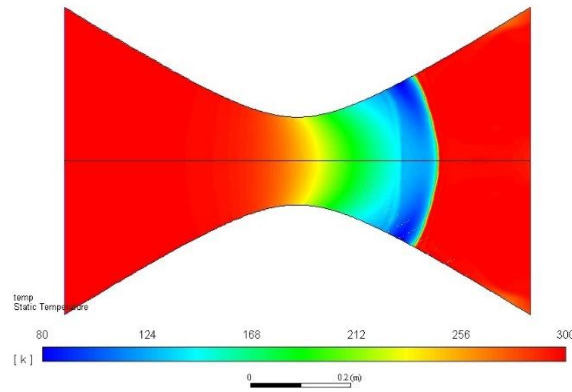


(c)

Not Available



(e)

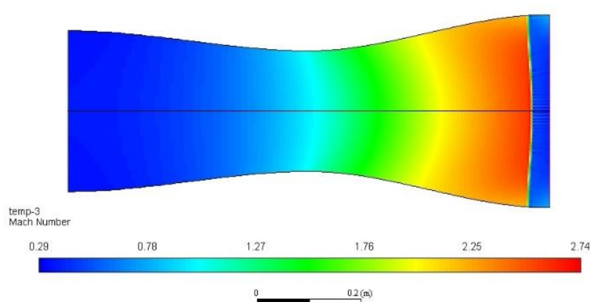


(f)

Figure 17: Temperature Contours

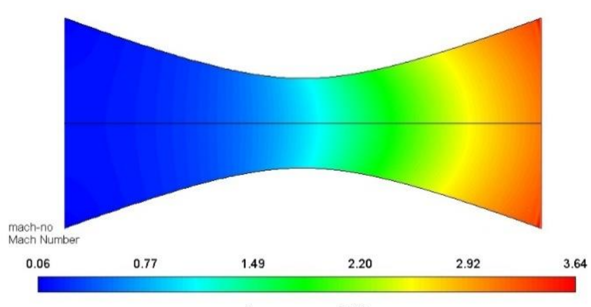
Same as in case I and II, the inlet temperature is kept as 300K. As soon as the flow passes through the throat of the nozzle, a gradual decrease in temperature is observed up to 120K where the shock is formed in design (a) as shown in Figure 17. The temperature contours for design (b) and (d) are not available as they were not compatible with these boundary condition. The temperature at the shock location of design (c, e, and f) is measured as 82, 112, and 80 respectively. The values for temperature is very low as compared to Case II for all designs.

4.3.3 Mach No Contours



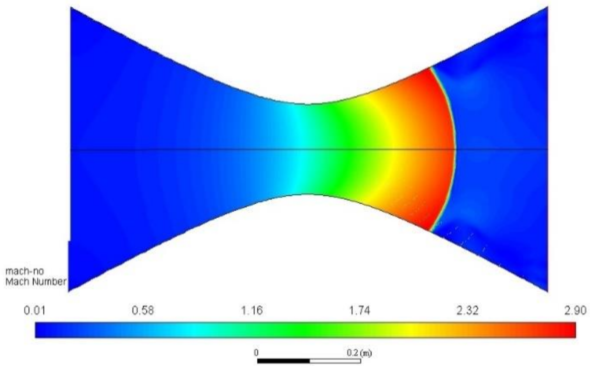
(a)

Not Available

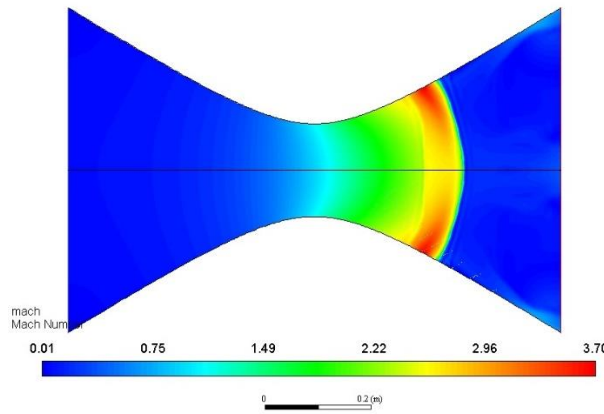


Not Available

(c)



(e)

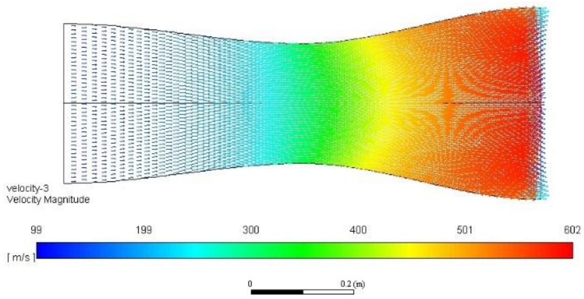


(f)

Figure 18: Mach No Contours

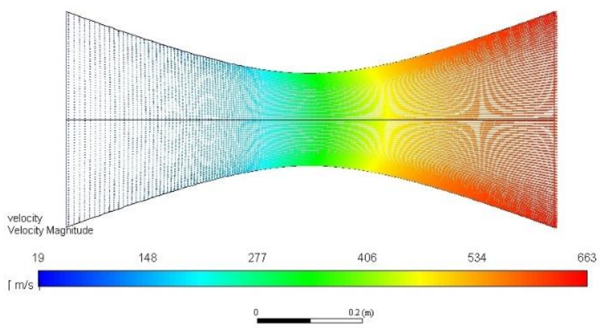
In Figure 18 it is shown that the Mach number is approximately one at the throat for all designs and highest at the shock front. The Mach no contours for design (b) and (d) are not available as they were not compatible with these boundary conditions. The viscosity causes a reduction in the momentum of the flow, as this loss is not considered in inviscid simulations. Therefore, the Mach number achieved is higher as compared to the standard k-ε turbulence model simulations (Case II) keeping the boundary conditions same in both the cases. The shock shape is straight in this case as there are no wall frictions. The Mach no for design (a, c, e, and f) is measured as 2.74, 3.64, 2.90, and 3.70 respectively which is higher than the Mach No observed in Case II.

4.3.4 Velocity Vectors



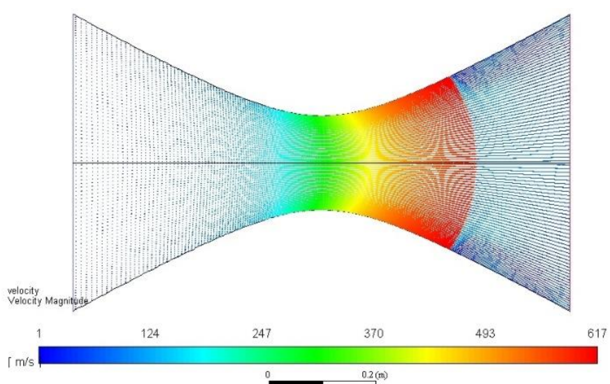
(a)

Not Available

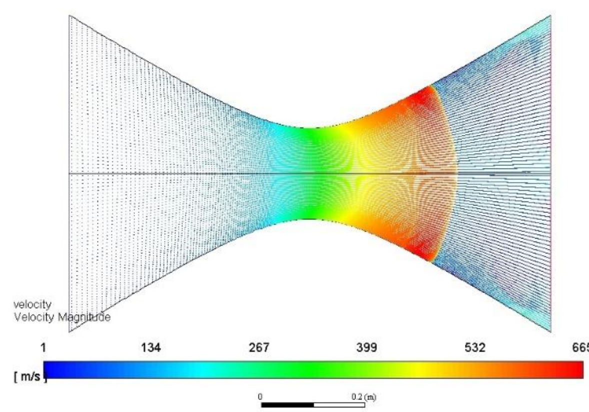


(c)

Not Available



(e)



(f)

Figure 19: Velocity Vectors

In Figure 19, the velocity vectors of flow showing increase in the velocity of the divergent section of the nozzle. The velocity vectors for design (b) and (d) are not available as they were not compatible with these boundary conditions. It is evident that there is no flow separation at the outlet of all designs. Hence, no vortex is formed or backflow is observed. The velocity gradients are small close to the wall. The axial velocity is more dominant as compared to Case II where the radial velocity was higher and velocity gradients were large due to wall friction and viscosity of the flow. The value of velocity is measured much higher as compared to Case II in all designs.

4.4 Comparative Analysis

A detailed analysis of all the designs with different input boundary conditions was carried out and the variation in pressure, temperature, Mach no, and velocity was studied. The designs (c, d, e, and f) were showing shock inside the nozzle even at high inlet pressure. The formation of shock inside the rocket motor nozzle is not desirable phenomena while designing a nozzle. Although higher divergence area give more acceleration to the flow but the sudden expansion causes shock. Therefore, these designs were ruled out for further comparison with the data available in the literature.

The design (a) and (b) can be considered for the analytical and comparative study with the data available in literature but the design (b) was not compatible for the inviscid flow inlet conditions as well as it was showing a minor flow separation even at high inlet pressure which causes reduction in thrust. Therefore, design (a) is selected amongst all the designs used for simulations. The design (a) is further validated with the already data available in the literature and analytical analysis is also carried out.

4.5 Validation of Results

The design (a) was selected based on the simulations carried out with different boundary conditions and turbulence models. The design (a) was further validated with data available in literature. The boundary conditions according to the literature with using the standard k-ε turbulence model was given to design (a) and CFD analysis was carried out. The results for variation in pressure, temperature, velocity, and Mach no was compared and analyzed. The Table 4 shows the boundary conditions taken from literature [33] and used for CFD analysis of our design (a).

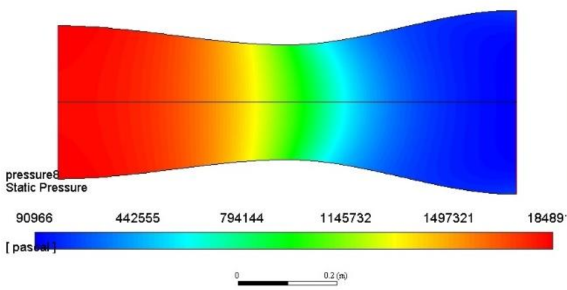
Table 3. Boundary Conditions taken from Literature

Pressure at Inlet	2 MPa
Temperature at Inlet	300 K
Pressure at Outlet	250 KPa
Temperature at Outlet	300

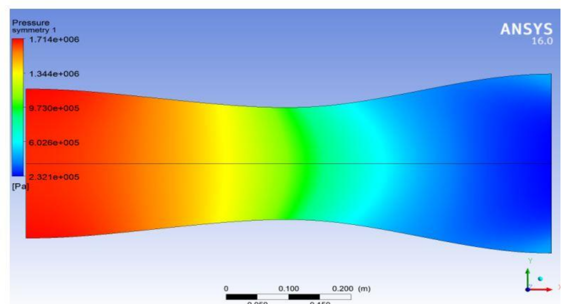
After giving the boundary conditions, the next step is to initialize the solution and run the calculations as per the desired number of iterations. Following are the results we have achieved and then same are compared with the literature results.

4.5.1 Pressure Contours and Graph

The same boundary conditions were applied on our model. The results for pressure contours and pressure graph are compared in Figure 20 and 21 respectively. The results shows that the pressure at the inlet is 1.8 MPa in our model and it was 1.71 MPa in the model which is taken as a reference. The flow passed through the nozzle throat area and expanded in diverging section of the nozzle. The value of pressure at the exit is 90KPa in our model and it is 232KPa in reference model. The flow has expanded close to the optimum value in our model. Same behavior is displayed in graphical form along the axis of the nozzle foe both the models in figure 21.

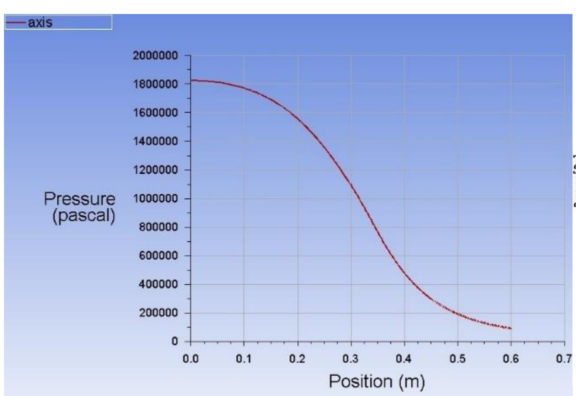


(This Model)

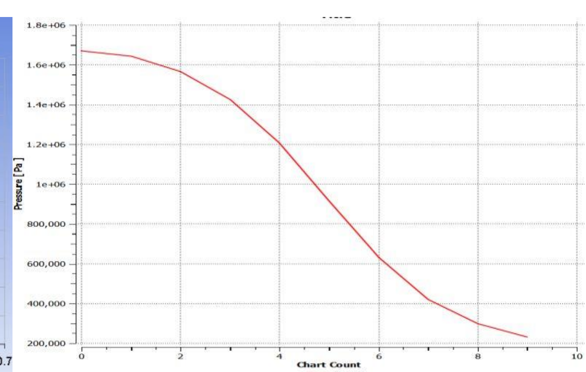


(Literature Data)

Figure 20: Comparison of Pressure Contours



(This Model)

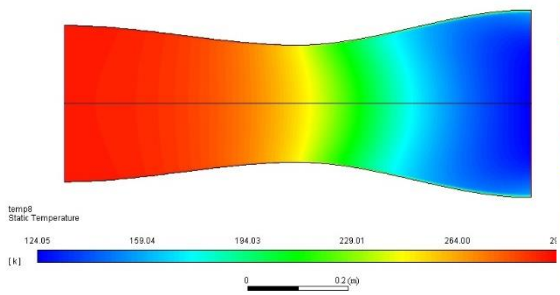


(Literature Data)

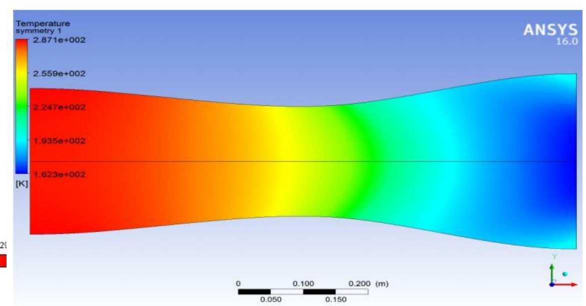
Figure 21: Comparison of Pressure Graphs

4.5.2 Temperature Contours and Graph

The results for temperature contours and temperature graph are compared in figure 22 and 23 respectively. It shows that the temperature at the inlet of our model is 298K and it is 287 K for the reference model. The higher the initial temperature the higher will be exit velocity as per equation 2. Therefore, we can say that our model will give better exit velocity as compared to the reference model. The temperature at the exit is measured as 124 K in our model and 162K in reference model. The graph shown in figure 23 also showing the same behavior.

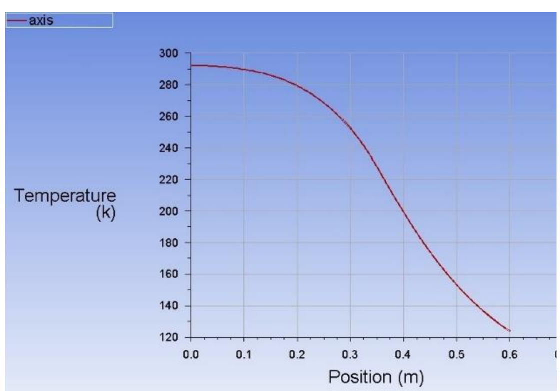


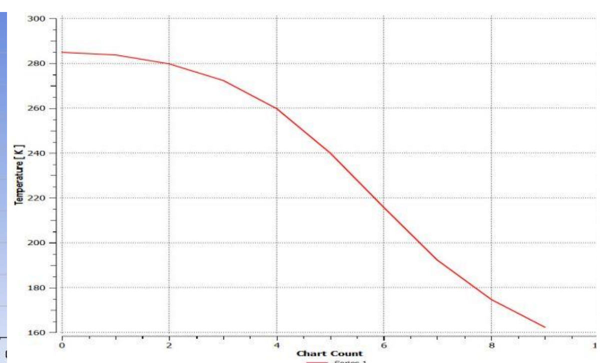
(This Model)



(Literature Data)

Figure 22: Comparison of Temperature Contours





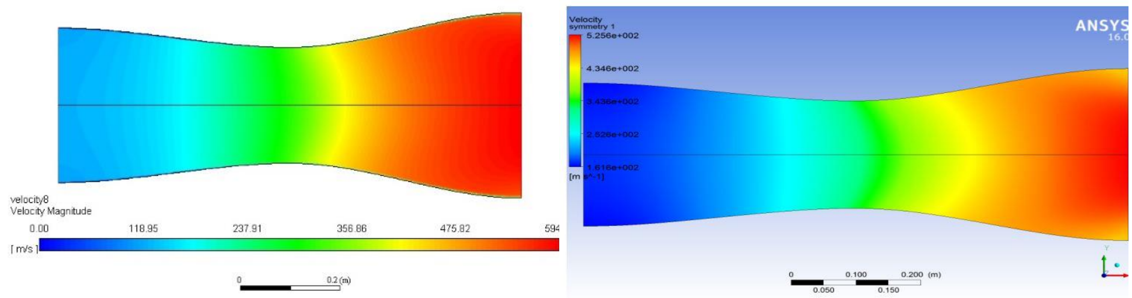
(Literature Data)

(This Model)

Figure 23: Comparison of Temperature Graphs

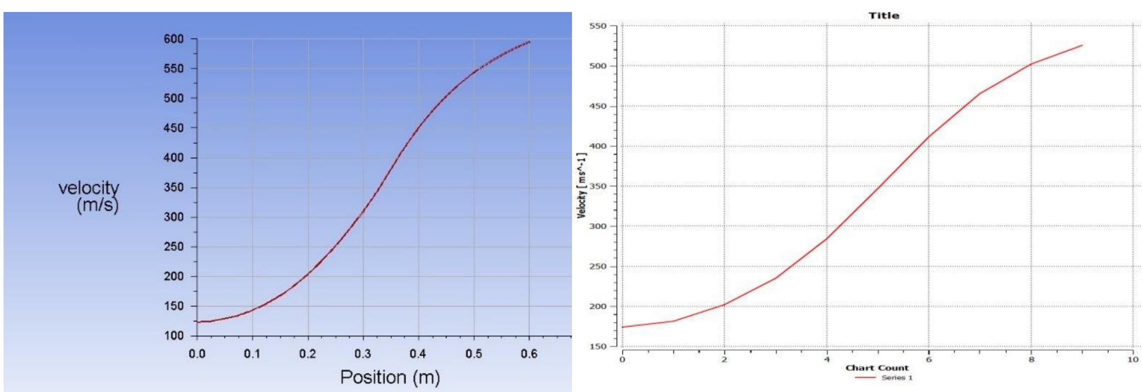
4.5.3 Velocity Contours and Graph

At last, the velocity was compared and the velocity contours are shown in figure 24 for both the models. It is evident from the contours that the velocity was very low initially in the converging section but it got accelerated after passing through the throat area. The exit velocity was measured as 594 m/s in our model and it was 525 m/s in the reference model. The same behavior of velocity is shown in graphical form along the axis of nozzle in figure 25.



(This Model) (Literature Data)

Figure 24: Comparison of Velocity Contours



(This Model) (Literature Data)

Figure 25: Comparison of Velocity Graphs

The table 4 will give the overall comparison of CFD analysis of both the models.

Table 4. Comparison of CFD Results

Factors	Our Model	Reference Model
Inlet Pressure	1.80 MPa	1.71 MPa
Outlet Pressure	90 KPa	232 KPa
Inlet Temperature	298 K	287 K
Outlet Temperature	124 K	162 K
Exit Velocity	594 m/s	525 m/s

Conclusions

A detailed study was conducted on the effect of inlet pressure variation on the Mach number and shock producing inside the nozzle using the standard k-ε turbulence model on 06 different models. The designing of the all nozzles was done in the ANSYS Design modeler. The meshing was carried out in the ANSYS MESH. Finally, the CFD analysis was carried out in ANSYS Fluent 18.1 for all the models under various cases. The main parameter which was variable in all the cases was the inlet pressure and its effect on the formation of shock, expansion of the gases at the exit and the variation in velocity at the exit was thoroughly studied. The pressure values were varied to check the position of the shock and the expansion of the product gases in the divergent portion of the nozzle in all the designs. The outlet pressure was kept constant to 100Kpa which is close to the atmospheric condition at sea level. The initial inlet pressure was set to 400 KPa and simulation were performed. No shock was observed inside the nozzle of design (a) and design (b). Moreover, the expansion of the gases was steady inside nozzle. With the increase in divergence area of the nozzle, the shock was observed in rest of the four designs. The flow separation at the exit of design (b) was also observed in this case.

The initial inlet pressure was gradually reduced. Simulations were carried out to study and analyze the behavior of exhaust flow inside the nozzle. In the second case, the Inlet pressure value was set to 240KPa, the shock was observed inside the nozzle. In this case, the expansion of the gasses was sudden, the shock was produced, and over-expanded nozzle behavior was observed in all the designs. The shape of the shock changed from curved to lambda as divergence area has increased.

In the third case, the flow was assumed to be inviscid. A shock was observed inside the nozzle in four designs. The design (b) and (d) were not compatible with these boundary conditions. It was evident that there were no wall frictions as the shape of shock is straight and it changed to curve in the design having largest divergence area. It can be concluded that the simulations using the standard k-ε turbulence model gives more realistic values than the simulation carried out using inviscid flow conditions.

After doing the analysis of all the design under three cases, it was found that the design (a) has shown the best performance in all the cases as compared to other design. It has displayed steady expansion without flow separation or shock producing inside nozzle in first case, the shock inside the nozzle and the shock close to the exit in case II and case III respectively. The performance of design (a) was validated by giving same the boundary conditions which are already available in the literature. The results of this model were compared to the literature results. It was evident that our model results were better than the design available in literature. The pressure ratio and the exit velocity are key parameters in calculating thrust. We have achieved better than the literature model results.

This study will enhance the indigenization in the field of rocketry in our country. Pakistan has recently launched a space program, moreover, strategic organization are working of number small and large scale rocket motor. This will give a start to new beginning in this field at an institute level and further studies in this field will be quit useful to the various organizations of Pakistan.

Future Recommendations

It is recommended that

1. This study was carried out in 2-D modeling, same can be carried out in 3-D modeling, which will be more realistic.
2. The standard wall treatment model was used in standard k-ε turbulence model, the enhanced wall treatment model can be used and compared with it.
3. The standard k-ω turbulence model can be used and results can be compared with this study.
4. This study was carried out only inside the nozzle, a study can be carried out inside the combustion chamber of the solid rocket motor. It will provide the burning pattern, pressure and temperature inside the chamber. These parameter can be used to design a complete small scale thruster at this Institution.

References

1. Sutton, G.P. and O. Biblarz, *Rocket propulsion elements*. 2016: John Wiley & Sons.
2. Hossain, M.S., M.F. Raiyan, and N.H. Jony, *Comparative study of supersoni c nozzles.* International Journal of Research in Engineering and Technology, 2014. **3**(10).
3. Biju Kuttan, P. and M. Sajesh, *Optimization of divergent angle of a rocket engine nozzle using computational fluid dynamics.* Optimization, 2013. **2**(2): p. 196-207.
4. Jayakumar, V., et al., *INVESTIGATION OF THERMAL CHARACTERISTICS IN SOLID ROCKET NOZZLE WITH INSULATE USING CAD/CAE.* International Journal of Pure and Applied Mathematics, 2018. **119**(7): p. 443-456.
5. Sofyan, S. and V. Wuwung, *RX-320 Rocket Static Pressure Combustion Chamber Prediction and Validation by Using Invers Method.* Jurnal Teknologi Dirgantara, 2018. **16**(1): p. 45-58.
6. Deshpande, N.D., et al., *theoretical and CFD Analysis of De-Laval Nozzle.* International Journal of Mechanical And Production Engineering, 2014. **2**(4): p. 2320-2092.
7. Khan, S.A., et al., *Analysis of Area Ratio In a CD Nozzle with Suddenly Expanded Duct using CFD Method.* Akademia Baru 2019**11**(5): p. 61-71.
8. Maddu, Y.R., S. Saidulu, and M.A. Jabiulla, *Design and Fluid Flow Analysis of Convergent-Divergent Nozzle.* 2018.
9. AABID, A. and S. KHAN, *NUMERICAL ANALYSIS OF CONVERGENT-DIVERGENT NOZZLE USING FINITE ELEMENT METHOD.* International Journal of Mechanical and ProductionEngineering Research and Development (IJMPERD), 2018. **8**(6): p. 373-381.
10. Akhtar, N. and S. Khan, *Numerical Simulation of Suddenly Expanded Flow at Mach 2.2.* International Journal of Engineering and Advanced Technology (IJEAT), 2019. **8**(3): p. 457-462.
11. Natta, P., V.R. Kumar, and Y.H. Rao, *Flow analysis of rocket nozzle using computational fluid dynamics (Cfd).* International Journal of Engineering Research and Applications, 2012. **2**(5): p. 1226-1235.
12. Belega, B.-A. and T.D. Nguyen. *Analysis of flow in convergent-divergent rocket engine nozzle using computational fluid dynamics*. in *International Conference of Scientific Paper*. 2015.

13. Pandey, K. and A. Singh, *CFD analysis of conical nozzle for mach 3 at various angles of divergence with fluent software.* International Journal of Chemical Engineering and Applications, 2010. **1**(2): p. 179.
14. Ramji, V., R. Mukesh, and I. Hasan. *Design and Numerical Simulation of Convergent Divergent Nozzle*. in *Applied Mechanics and Materials*. 2016. Trans Tech Publ.
15. Khan, S.A. and A. Aabid, *CFD Analysis of CD Nozzle and Effect of Nozzle Pressure Ratio on Pressure and Velocity For Suddenly Expanded Flows.* International Journal of Mechanical and Production Engineering Research and Development, 2018. **8**: p. 1147-1158.
16. SUDHAKAR, B.N., et al., *Modeling and simulation of Convergent-Divergent Nozzle Using Computational Fluid Dynamics.* International Research Journal of Engineering and Technology (IJRET), 2016. **3**(8).
17. Satyanarayana, G., C. Varun, and S. Naidu, *CFD analysis of convergent-divergent nozzle.* Acta Technica Corviniensis-Bulletin of Engineering, 2013. **6**(3): p. 139.
18. Prafulla, M.B.K., D.V.C. Babu, and S.P.G. Rao, *CFD Analysis of Convergent-Divergent Supersonic Nozzle.* International Journal of Computational Engineering and Research, India, 2013. **3**(5): p. 5-15.
19. Mohan Kumar, G., D.X. Fernando, and R.M. Kumar, *Design and Optimization of De Lavel Nozzle to Prevent Shock Induced Flow Separation.* Advances in Aerospace science and Applications, 2013. **3**(2).
20. Swaroopini, A.S., M.G. Kumar, and T.N. Kumar, *Numerical Simulation and Optimization of High Performance Supersonic Nozzle at Different Conical Angles.* International Journal of Research in Engineering and Technology (IJRET), 2015. **4**(09).
21. Ahsan, M., *Numerical analysis of friction factor for a fully developed turbulent flow using k–ε turbulence model with enhanced wall treatment.* Beni-Suef University journal of basic and applied sciences, 2014. **3**(4): p. 269-277.
22. Najar, N.A., D. Dandotiya, and F.A. Najar, *Comparative analysis of k-ε and spalart-allmaras turbulence models for compressible flow through a convergent-divergent nozzle.*
23. Rolander, N., et al., *An approach to robust design of turbulent convective systems.* Journal of Mechanical Design, 2006. **128**(4): p. 844-855.

24. Rao, G.R., U. Ramakanth, and A. Lakshman, *Flow analysis in a convergent-divergent nozzle using CFD.* International Journal of Research in Mechanical Engineering, 2013. **1**(2): p. 136-44.
25. Pansari, K. and S. Jilani, *Numerical Investigation of the Perfomance of Convergent Divergent Nozzle.* International Journal of Modern Engineering Research (IJ MER), 2013. **3**(5): p. 2662-2666.
26. KiranKopelli, J.R.R., M. Sudhakar, and G.B. Rao, *COMPARATIVE CFD ANALYSIS OF THREE FLOW VISCOUS MODELS FOR AIR FLOWING AT HIGH SPEED THROUGH A CONVERGENT-DIVERGENT NOZZLE.*
27. Reji, A.K., et al., *Simulation and validation of supersonic flow through a convergent divergent nozzle.* International Journal of Pure and Applied Mathematics, 2018. **119**(12): p. 2135-2142.
28. Schmidt, D.P., C.J. Rutland, and M.L. Corradini, *A fully compressible, two-dimensional model of small, high-speed, cavitating nozzles.* Atomization and sprays, 1999. **9**(3).
29. Srinivas, G. and S.R. Potti. *Numerical Simulation of Rocket Nozzle*. in *Advanced Materials Research*. 2014. Trans Tech Publ.
30. Mueller, T., W.P. Sule, and C.R. Hall, *Characteristics of separated flow regions within altitude compensating nozzles*. 1971: University of Notre Dame.
31. Dhital, D., et al., *A review of flaws and damage in space launch vehicles: Motors and engines.* Journal of Intelligent Material Systems and Structures, 2014. **25**(5): p. 524-540.
32. Khan, A. and T. Shembharkar, *Viscous flow analysis in a convergent-divergent nozzle.* 2008.
33. Patel, M.S., S.D. Mane, and M. Raman, *Concepts and CFD Analysis of De-Laval Nozzle.* International Journal of Mechanical Engineering and Technology (IJMET), 2016.