Design and Analysis of Injector with Different Configurations for Liquid Rocket Engine

A J Sriganapathy[1] , Varunavan P[2] , Ramana.S[3], Praveen Kumar R[4] , M.Ranjani[5]
[1]*Assistant Professor- Department of Aeronautical Engineering, Mahendra Institute of Engineering and Technology, Tiruchengode, Namakkal, Tamilnadu, India*
[2,3,4,5] *UG student of Aeronautical Engineering, Mahendra Institute of Engineering and Technology, Tiruchengode, Namakkal, Tamilnadu, India*

ABSTRACT
The performance of the rocket is determined by atomization, mixing and combustion process. Injectors are used to control these processes. Suitable method should be followed, so that the performance should be increased. The injector implementation in rockets determines the performance of the nozzle that can be achieved. Injectors can be as simple as a number of small diameter holes arranged in carefully constructed patterns through which the fuel and oxidizer travel. The performance of an injector can be improved by either using a superior propellant combustion, increasing the mass flow rate, positioning the angle of the hole or by reducing the size & increasing the number of orifices on the injector plate. In here the position is varied for different angles for injectors to improve the performance of the rocket. The injector is modeled using the Solidworks and a two dimensional combustion analysis to be carried out using ANSYS Fluent for different cases.

1. INTRODUCTION

A rocket is a missile, spacecraft, aircraft or other vehicle that obtains thrust from a rocket engine. Rocket engine exhaust is formed entirely from propellant carried within the rocket before use. Rocket engines work by action and reaction and push rockets forward simply by expelling their exhaust in the opposite direction at high speed, and can therefore work in the vacuum of space. In fact, rockets work more efficiently in space than in an atmosphere. Multistage rockets are capable of attaining escape velocity from Earth and therefore can achieve unlimited maximum altitude. Compared with air breathing engines, rockets are lightweight and powerful and capable of generating large accelerations. To control their flight, rockets rely on momentum, airfoils, auxiliary reaction engines, gimbaled thrust, momentum wheels, deflection of the exhaust stream, propellant flow, spin, and/or gravity. Chemical rockets are the most common type of high power rocket, typically creating a high speed exhaust by the combustion of fuel with an oxidizer. The stored propellant can be a simple pressurized gas or a single liquid fuel that disassociates in the presence of a catalyst (monopropellants), two liquids that spontaneously react on contact (hypergolic propellants), two liquids that must be ignited to react, a solid combination of fuel with oxidizer (solid fuel), or solid fuel with liquid oxidizer (hybrid propellant system). Chemical rockets store a large amount of energy in an easily released form, and can be very dangerous. However, careful design, testing, construction and use minimizes risks.

2. PROBLEM DEFINITION & OBJECTIVE
Problem description:
The fuel injector used in the liquid rocket engine should have a capability of better performance in terms of all aspects. The injector is a main part in the rocket engine; the injector must be chosen to improve the performance. The injectors have different configurations like doublet, triplet injectors. The primary focus is to improve the performance by concentrating the fuel injectors design.

Objective: The objective of the project is to improve the performance of the engine by implementing the suitable injector for the rocket engine. Engine performance mentioned below has to be improved by selecting a suitable design.

- ☐ Fuel consumption
- ☐ Fuel rich flammability limits
- ☐ Good flammability.

3. DESIGN OF FUEL INJECTOR

3.1 Modeling Tool:

2D design tools and Solid Works have fundamentally different approaches. In 2D design tools, you design in a 2D environment. In Solid Works, you design in a 3D environment, and you create 2D drawings based on the 3D model.

3.2 Types of Models

Computer-Aided Design software packages handle models in the following ways:
- 2D drawings
- Wireframe models
- Surface models (organic shapes)
- Solid models (mechanical parts and assemblies)
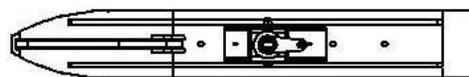
2D Drawings
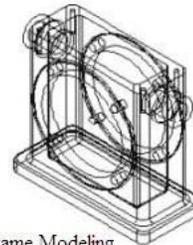
Wireframe Modeling
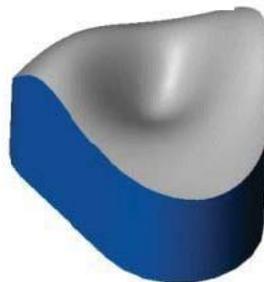
Surface Modeling

Solid Modeling

Solid Works creates solid models, but it can also import, create, and manipulate surfaces, view models in wireframe mode, and generate 2D drawings from the 3D solid models. Scan to 3D tools, available in Solid Works Premium, import mesh and point cloud data from which you can create surfaces and solid models.

3.3 Sketching versus Drawing:

In Solid Works, drawings are the 2D documents that you create from 3D part or assembly models. The tools that are considered drawing tools in 2D CAD programs are sketching tools in Solid Works. When developing models in Solid Works, you sketch geometric entities (such as rectangles and circles) as the

basis for solid features (such as extrusions, revolves, and cuts). You can sketch entities approximately, and then dimension the entities exactly.

3.4 Feature-based Models

Just as an assembly consists of individual parts, a Solid Works part consists of individual features. The first feature you create in a part is the base. This feature is the basis on which you create the other features. The base feature can be an extrusion, a revolve, a sweep, a loft, thickening of a surface, or a sheet metal flange. However, most base features are extrusions.

3.5 Configurations

Configurations in Solid Works allow you to create multiple variations of a part or assembly model within a single document. Configurations are a convenient way to develop and manage families of models with different dimensions, components, or other parameters. You can create configurations manually, or you can use a design table to create multiple configurations simultaneously. Design tables provide a convenient way to create and manage configurations in a worksheet. You can use design tables in both part and assembly documents.

Modeling in 3D helps you stay organized and in touch with the real world you're designing for. With Solid Works, increased speed and accuracy free your design team to be more creative, so you can design products smarter, faster and better.

3.6 Design Parameters:

Three Stages
1. 14 Rings
2. 11 Rings
3. 4 Rings

Ring Depth =1.367 cm

Wall Gap = 1.778 cm

Angle of First stage = 0 degree

Angle of Second stage = 28.2 degree Angle of Third stage = 20 degree Fuel = Kerosene;

Oxidizer = Liquid oxygen

Injection Velocity for Fuel = 17.07 m/s; Injection Velocity Oxidizer = 40.54 m/s

Total Holes = 2090 holes

Mixture Ratio = 2.27

Vacuum Specific Impulse = 269 s

Vacuum Thrust = 8,000,000 N

Ring No.	Fuel / Oxidizer	No. of Holes	Hole Diameter	Ring Diameter
1	F	18	0.199	1.915
2	O	18	0.242	3.116
3	F	18	0.281	4.245
4	O	18	0.209	5.246
5	F	24	0.209	7.256
6	O	40	0.281	8.296
7	F	48	0.242	9.368
8	O	48	0.281	10.868
9	F	64	0.242	12.06
10	O	64	0.281	13.236

11	F	72	0.242	14.34
12	O	72	0.281	15.466
13	F	88	0.242	16.314
14	O	88	0.281	17.706
15	F	72	0.209	18.661
16	O	90	0.209	23.446
17	F	112	0.281	24.426
18	O	128	0.242	25.546
19	F	128	0.281	26.666
20	O	144	0.242	27.798
21	F	144	0.281	28.806
22	O	144	0.242	30.026
23	F	144	0.281	31.146
24	O	160	0.242	32.266
25	F	160	0.281	33.385
26	O	176	0.242	34.606
27	F	176	0.281	35.626
28	O	192	0.209	36.746
29	F	192	0.228	37.766
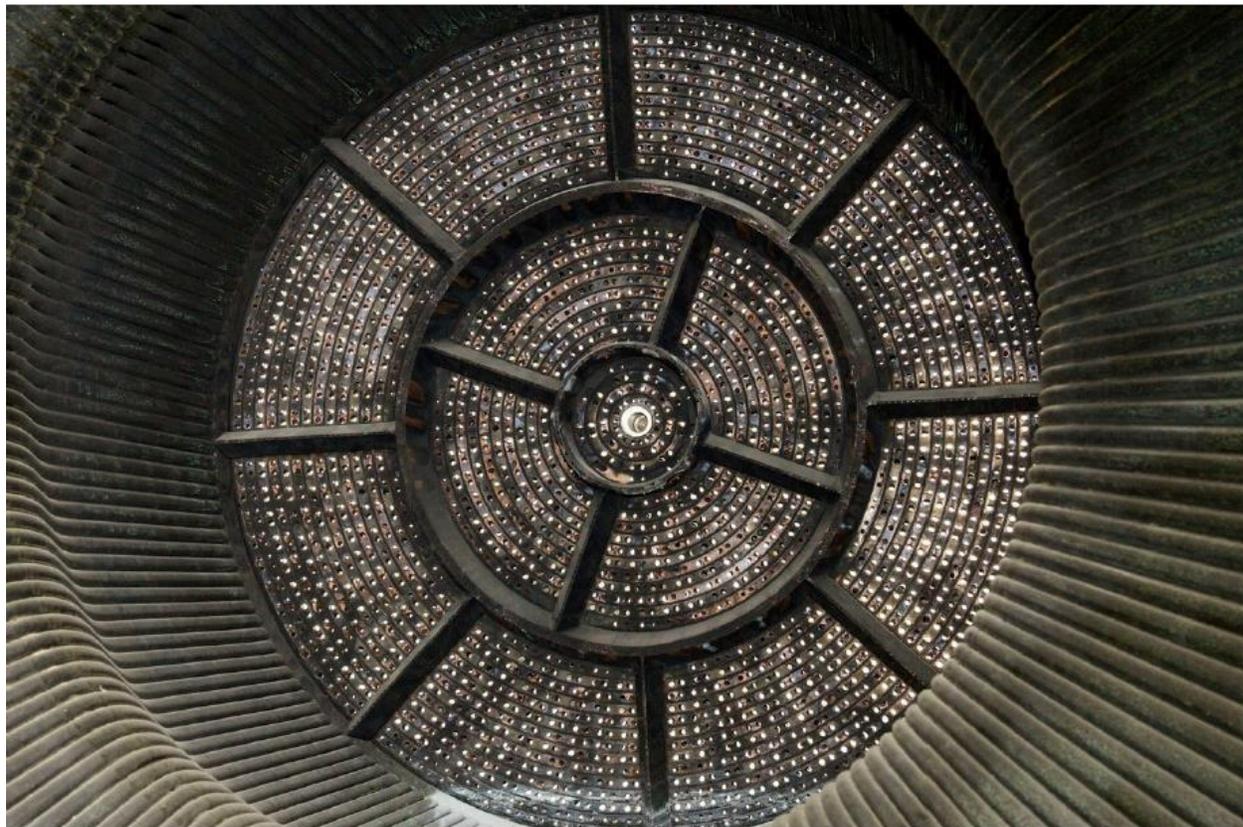
Fuel injector fabricated model

Fuel injector with injector tubes

Based on the design parameters given above the fuel injector is designed using Solidworks
2018 and the model is shown in below. The injector is designed for five different angles. The angles are
of 30°, 40°, 45°, 50° and 60°. The modeled images of these configurations are given in below.
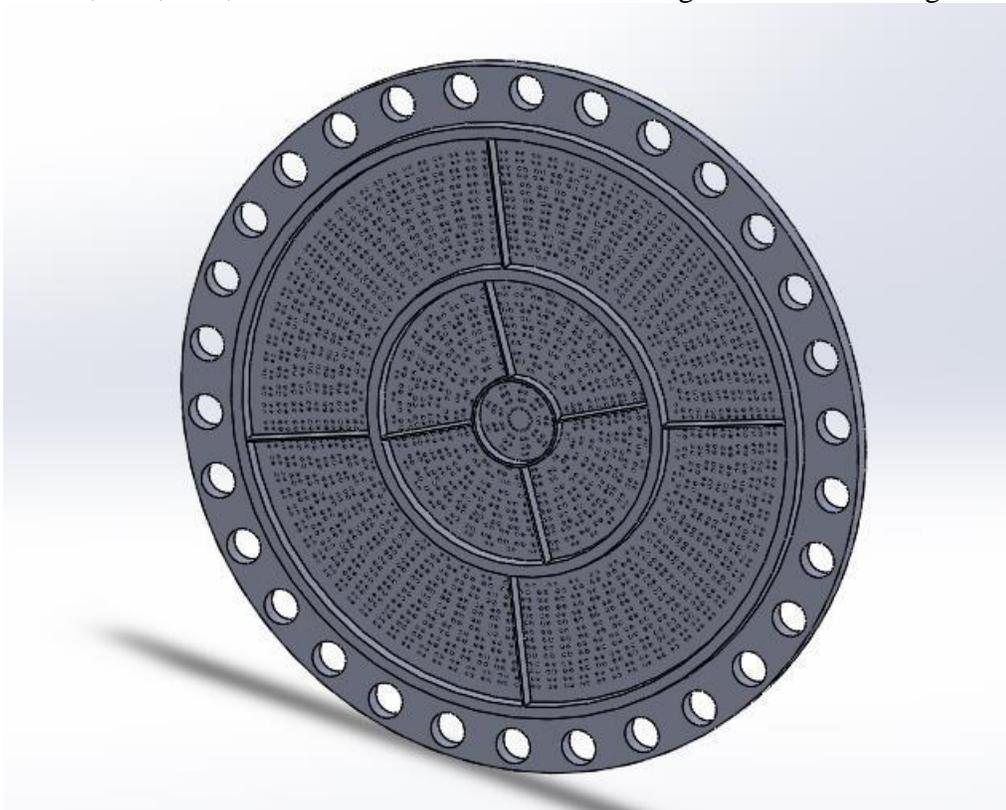
3D model of injector
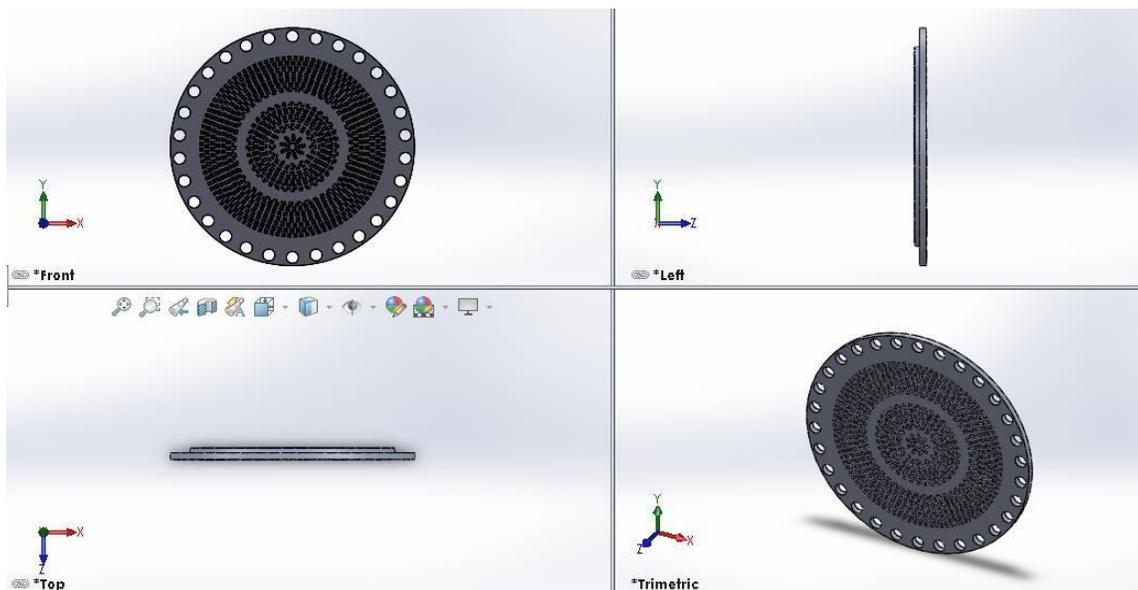
Three view of injector design model

4. NUMERICAL ANALYSIS OF FUEL INJECTOR
4.1 CFD:
Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. Even with high-speed supercomputers only approximate solutions can be achieved in many cases.

4.2 OUTLINE OF COMPUTATIONAL FLUID DYNAMIC PROCESS
Computational Fluid Dynamic codes are structured around the numerical algorithms that can tackle fluid flow problems. All the CFD codes available in the market have three basic elements which divide the complete analysis of the numerical experiment to be performed on the specific domain or geometry. The three basic elements are
i.   Pre-processor ii.   Solver and
iii.   Post-Processor

4.3 MESHING AND PRE-PROCESSING
The pre-processing of the CFD process consists of the input of a flow problem by means of user-friendly programs or software and the subsequent transformation of this input into a form is made suitable to use by the solver. The pre-processor is the link between the user and the solver. The user activity at the pre-processing stage of the CFD process involves the following:
1) Definition of Geometry or region of Interest: This process involves several computer aided design (CAD) software like CATIA, Solidworks, Pro-E and much more. By the help of CAD software, the topology of the fluid flow region of interest is defined. This software plays a major part of the design and optimization process in research analysis.
2) Grid Generation or Meshing: Since the CFD process is a numerical approximation method using finite volume method, the given domain or region of interest needs to be divided

into several structured elements. All the elements or cells are connected to each other through nodes to form the required region of flow. For this purpose, special meshing or grid generation software like Ansys, GAMBIT and T-grid are used. This stage is the key element in the CFD finite volume numerical simulation and it also contributes to the accuracy of the final results.

3) Definition of Fluid properties: Every fluid domain or surface has its own distinct property. The properties of the fluid used in the CFD domain or region of interest are defined at this stage of the CFD Process. Usually the CFD code software has this facility.

4) Boundary Conditions: Every different setup of the CFD domain needs to have an initialization, which is fulfilled by the boundary conditions input. The CFD code usually has this facility to define the boundary conditions of the CFD problem, where each cells at specific boundary are given finite values.

4.4 NUMERICAL SOLVER

The numerical solver is the key elements of the CFD process and covers the major part of the CFD process. In the current market, the solvers usually use three distinct ways of calculating the solutions, namely, the finite difference method, finite element method and the finite volume method.

The finite difference and element method are usually suitable for stress and structure analysis and does not suite the requirements of the CFD process. The finite volume method is the most suitable method for the CFD process. As the name implies, finite volume method is the numerical algorithm calculation process involving the use of finite volume cells. The steps involved in this solving process are usually carried out in the following sequence:

1. Formal integration of the governing equations of fluid flow over all the control volumes or finite volumes of the solution domain.

2. The conversion of the integral forms of the equations into a system of algebraic equations.

3. Calculations of the algebraic equations by an iterative method.

4.5 POST PROCESSOR

The post processor is the last phase of the CFD process which involves data visualization and results analysis of the CFD process. This phase uses the versatile data visualization tools of the CFD solver to observe the following results of the simulation:

1. Domain geometry and Grid display

2. Vector plots

3. Line and shaded contour plots

4. 2D and 3D surface plots

5. Particle tracking

6. XY plots and graphs of results

In this projectthe outline given above is followed to complete the analysis of each CFD

simulation performed. The CFD process starts with defining the geometry using the CAD software

'Solidworks' and then it is followed by the meshing software ANSYS, which is used to create the surface mesh accordingly.

After the surface mesh is created, the resulting surface mesh is imported to software, known as 'ANSYS-Fluent'. This software creates a hybrid mesh in the required domain of interest with the help of progressive boundary layer, tetra and Hex core cells.

4.6 GEOMETRY CREATION:
In this process geometry for the FUEL INJECTOR is drafted from the Solidworks CAD package to the design modeler. The drafted model of FUEL INJECTOR is shown in figure 4.1. Now a domain for the model is created for the analysis purpose. The domain with an inlet as a hemisphere is generated in here to obtain a better result.

Fig: 4.1 Model of fuel injector

By considering these parameters the fuel injector for analysis is created as shown in the figure 4.2.

Fig.4.2 Domain of the model

4.7 MESHING:

Different meshing characteristics were used for the domain. Although the methods were different, the same mixed configuration of triangular and quad core cells were used for the domain. The triangular shape surface mesh was used due to its proximity to changing curves and bends. These elements easily adjust to the complex bodies used in automobile and aerospace bodies. In the simulations, the vehicle component surfaces were discretized with triangular mesh elements. Cells of mixed cell type were used in the computational domain.

In initial a mesh with face meshing over the injector with is defined with the parameters defined asbelow:

Curvature angle   : 18°

Minimum Size   : 3.0934 x 10^{-2}mm Maximum Face Size   : 3.09340 mm Maximum Size   : 6.18680mm Growth Rate   : 1.2
Min. Edge Length   : 0.65270mm

Sizing over BWB   : Element size = 0.5 mm

After defining these parameters we have obtained a quite good mesh quality.

The meshed image is shown in figure 4.4. From this condition we have obtained a total number of nodes as 57859 and total number of elements as 57507.
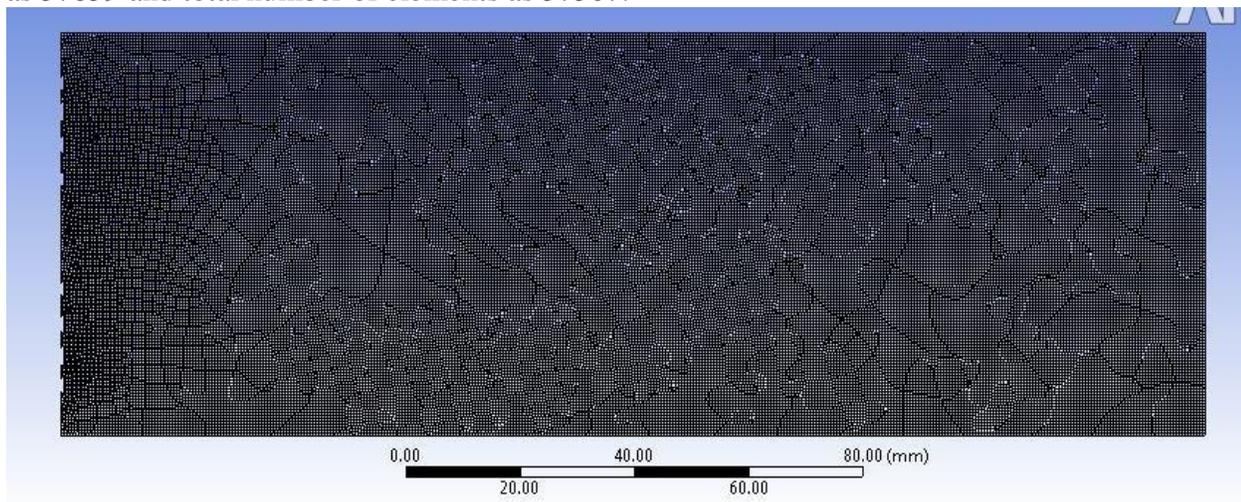
Fig.4.3 Mesh Model

The detailed mesh image is shown in the figure 6.4. It's the zoomed view of the mesh

which is located over the fuel injector model. The naming for the domain is as shown in the figure

4.5.
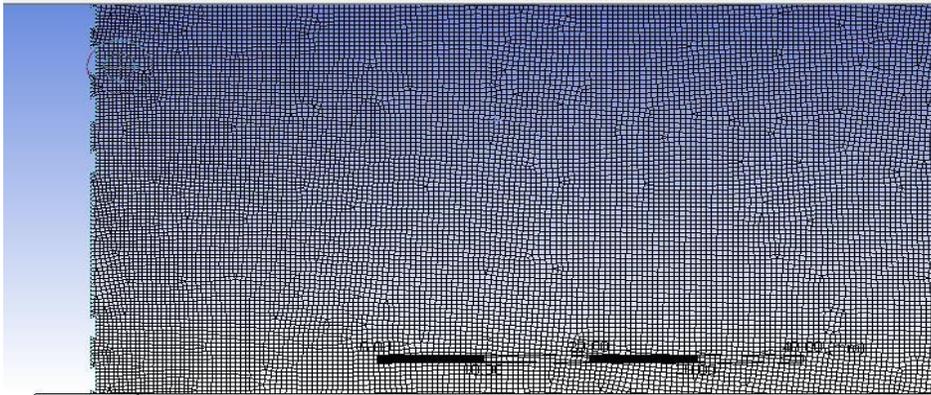
Fig.4.4 Mesh over the injector

The entire mesh statistics is as given below. These are the total data which we obtain after meshing the domain as per the parameters we defined earlier

| Zone	Number of faces
Inlet	62
Outlet	140
Wall	950
Interior	114213
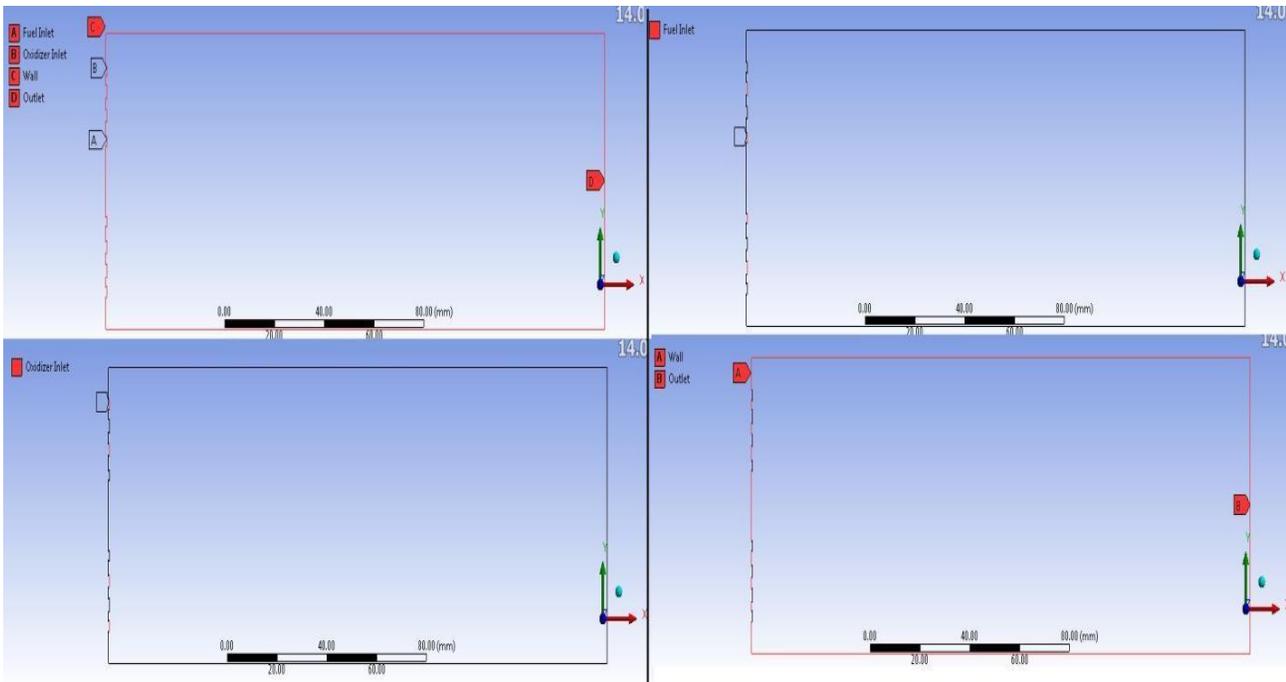
Fig. 4.5 Section Naming

4.8.SOLVER SETTINGS:

The problem of combustion numerical analysis requires the solver settings to be completed before starting the simulations. The solver setting includes type of solver (3D or 2D), the viscous model, boundary condition and solution controls. The inlet of the injector is indicated by the term 'Velocity inlet', while the outlet of the wind tunnel is termed as 'Pressure outlet'. The fluid properties were calculated taking into account the temperature and density of the average ambience condition. The solver settings and boundary condition for both the benchmark simulations are shown in the Table 6.1, Table 4.2 and Table 4.3.

Table.4.1 Solver setting

CFD Simulation	3D
Solver	
Solver	Pressure-Based
Space	3D
Formulation	Implicit
Time	Steady
Velocity Formulation	Absolute
Gradient Option	Cell-Based
Porous Formulation	Superficial Velocity

Table 4.2 Viscous model and Turbulence model settings

Viscous Model	
Turbulence Model	k-ε (2 eqn)
k-epsilon Model	Standard
Near-Wall Treatment	Standard Wall Functions
Operating Conditions	Ambient
Total Kinetic Energy Prandtl Number	1
Total Dissipation Rate Prandtl Number	1.3
Energy	ON
Radiation	Discrete Ordinates
Species	Non – Premixed Combustion
Stream rich limit	0.5
Fuel	RP-1 (CH2)
Oxidizer	O2

Table 4.3 Boundary condition

	Boundary Conditions	
Velocity Inlet-1 (Fuel)	Magnitude (Measured normal to Boundary)	17.07 m∕s (constant)
	Pressure	20×10^6 Pa
	Temperature	280 K
	Turbulence Specification Method	Intensity and Viscosity Ratio
	Intensity ratio	5
	Viscosity ratio	5
Velocity Inlet-2 (Oxidizer)	Magnitude (Measured normal to Boundary)	40.54m∕s (constant)
	Pressure	20×10^6 Pa
	Temperature	260 K
	Turbulence Specification Method	Intensity and Viscosity Ratio
	Intensity ratio	5
	Viscosity ratio	5
Pressure Outlet	Gauge Pressure magnitude	101325 Pascal
	Gauge Pressure direction	normal to boundary
	Turbulence Specification Method	Intensity and Viscosity Ratio
	Backflow Intensity	5
	Backflow Viscosity	5
Wall Zones	No Slip	

The simulation results obtained for the Fuel Injector model are discussed in the next chapter.

5. RESULTS AND DISCUSSION
The simulation results obtained for the Fuel Injector model is as given below. The contours of the pressure, velocity, temperature, and Mean mixture fraction are shown in below. The numerical values as depicted in the below table are interpreted from the contours.

Table.5.1

Sl.No	Angle	Velocity (m/s)	Mean mixture fraction	Temp (K)	Pressure (Pa)
1	40	132	0.82	824	1.13×10^5
2	42.5	143	0.75	711	1.1×10^5
3	45	134	0.8	826	9.7×10^4
4	47.5	130	0.75	843	8.89×10^4

5	50	165	0.74	708	8.65×10^4

5.1 Velocity Distribution
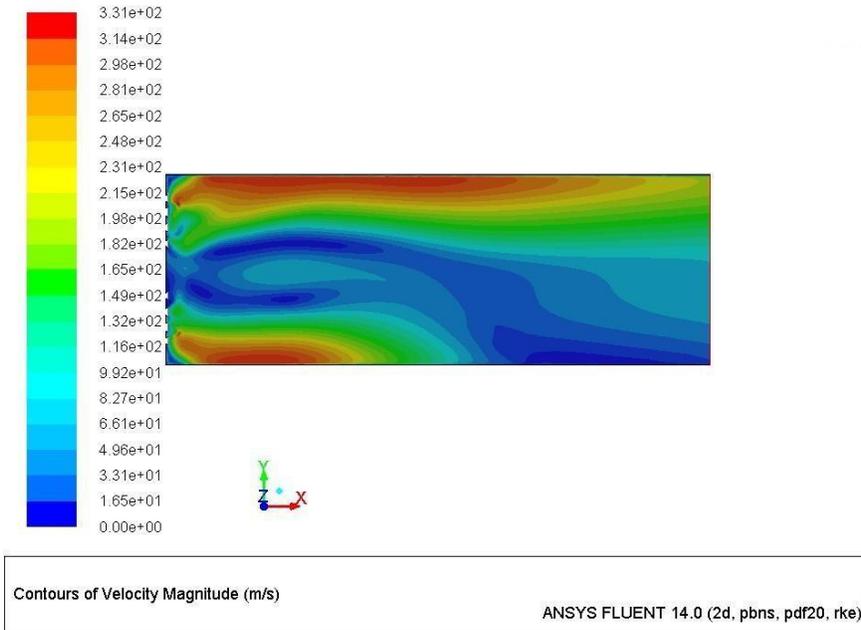
Figure.5.1 Velocity contour for 40°injection
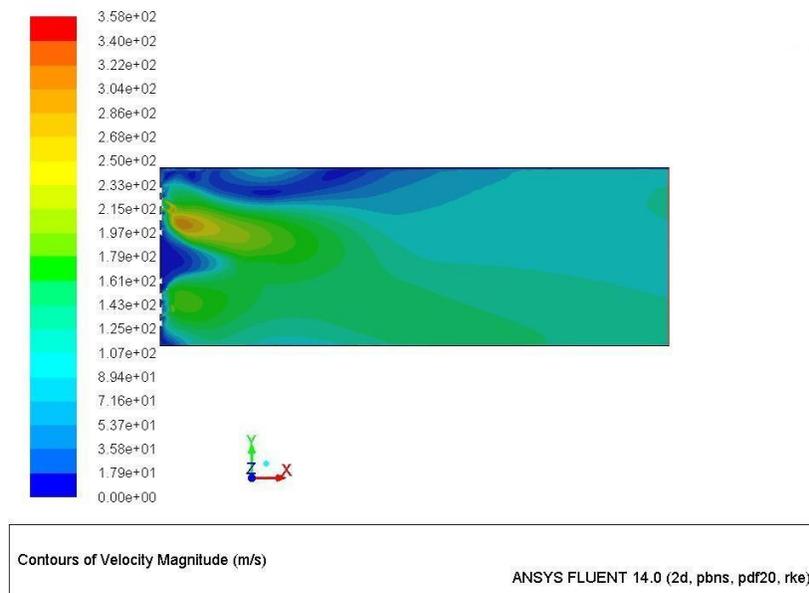
Figure 5..2 Velocity contour for 42.5°injection
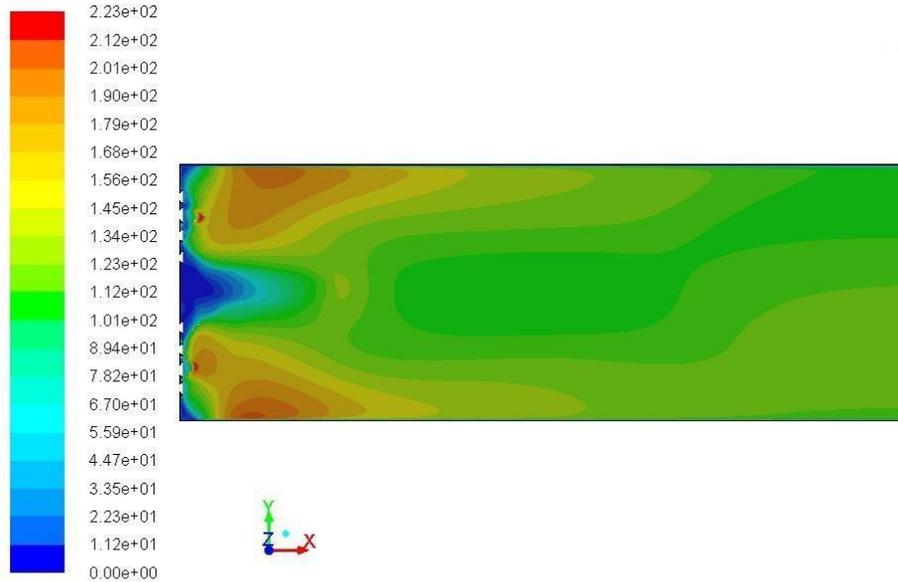
Figure.5.3 Velocity contour for 45°injection
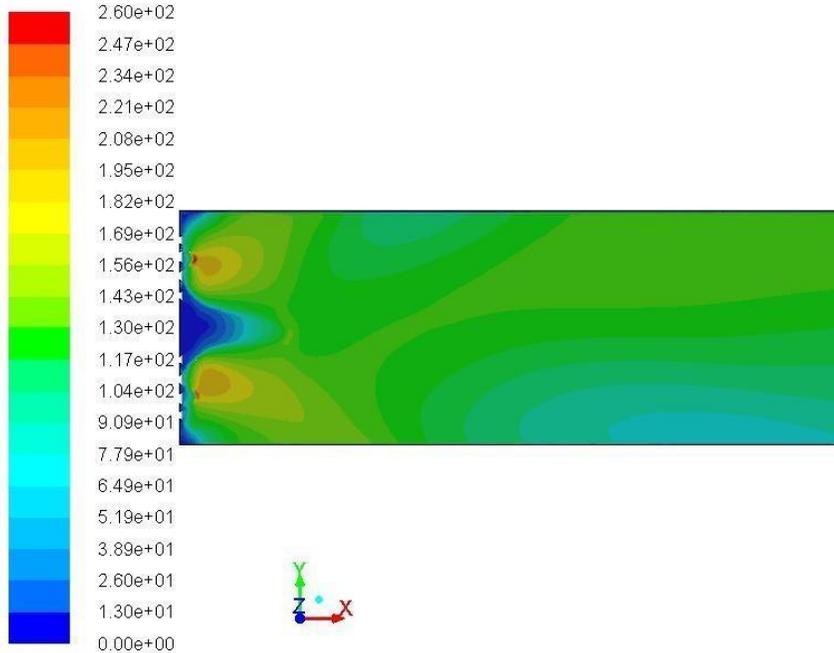

Figure.5.4 Velocity contour for 47.5°injection
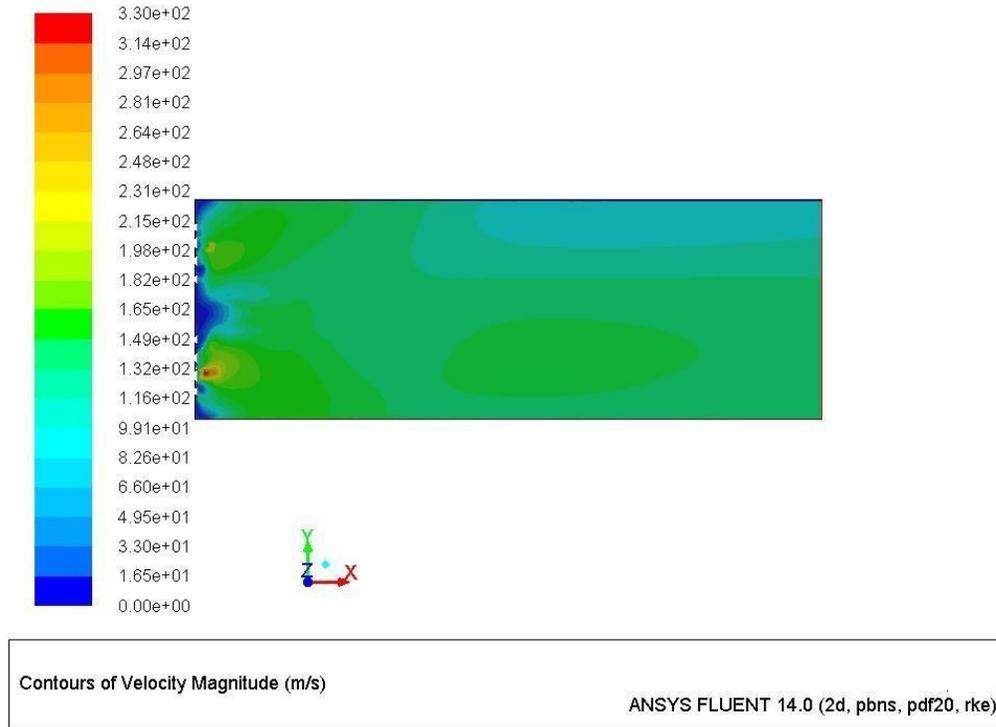
Figure.5.5 Velocity contour for 50°injection

The velocity distributions in Fuel injector for various angles are given from figure 1 to 5. In figure 1, we have the Velocity of 132 m/s at angle of 40^0. As well as, in figure 2, we have the Velocity of 143 m/s at angle of 42.5^0. In figure 3, we have the Velocity of 134 m/s at angle of 45^0. In figure 4, we have the Velocity of 130m/s at angle of 47.5^0. Finally, in figure 5, we have the Velocity of 165 m/s at angle of 50^0. By comparing all velocities with respect to various angles of attacks, we have maximum velocity of 165 m/s at angle of 50^0

5.2. Mean mixture Fraction

The Mean mixture fractions in Fuel injector for various angles are given from figure 6 to 10. In figure 6, we have the mean mixture fraction of 0.82 at angle of 40^0. As well as, in figure 7, we have the mean mixture fraction of 0.75 at angle of 42.5^0. In figure 8, we have the mean mixture fraction of 0.8 at angle of
45^0. In figure 9, we have the mean mixture fraction of 0.75 at angle of 47.5^0. Finally, in figure 10, we have the mean mixture fraction of 0.74 at angle of 50^0. By comparing all mean mixture fractions with respect to various angles of attacks, we have minimum mean mixture fraction of 0.74 at angle of 50^0
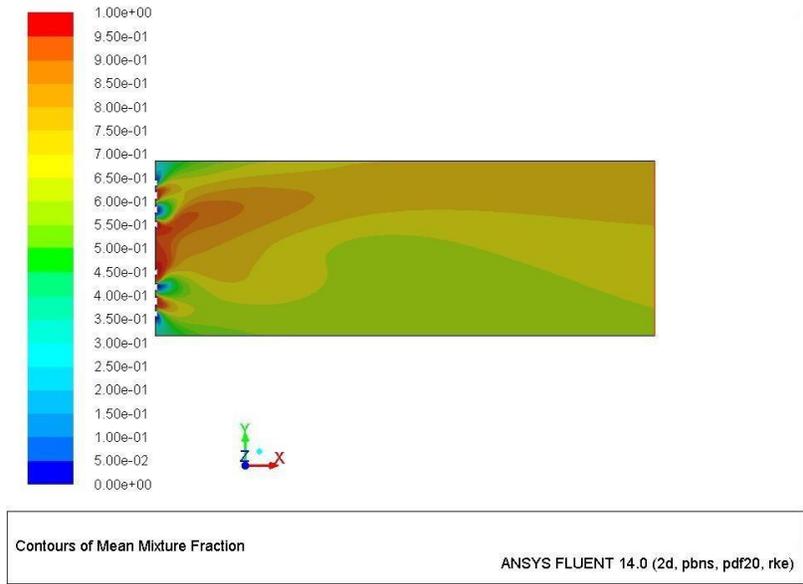
Figure.5.6 Mean Mixture Fraction contour for 40°injection
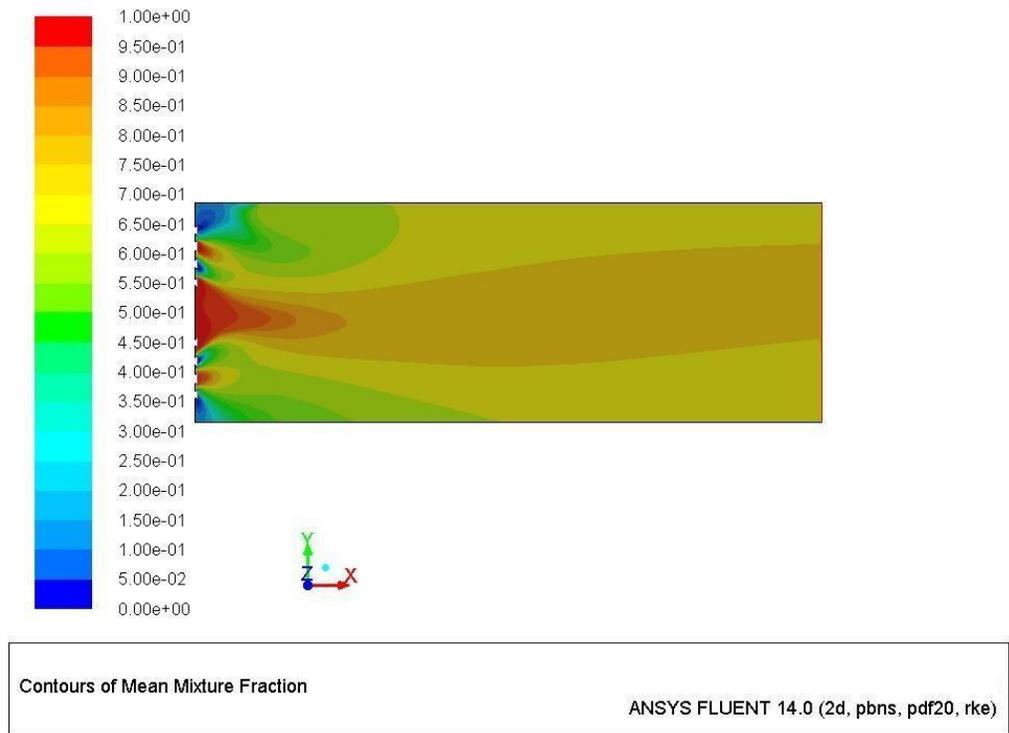
Figure.5.7 Mean Mixture Fraction contour for 42.5°injection
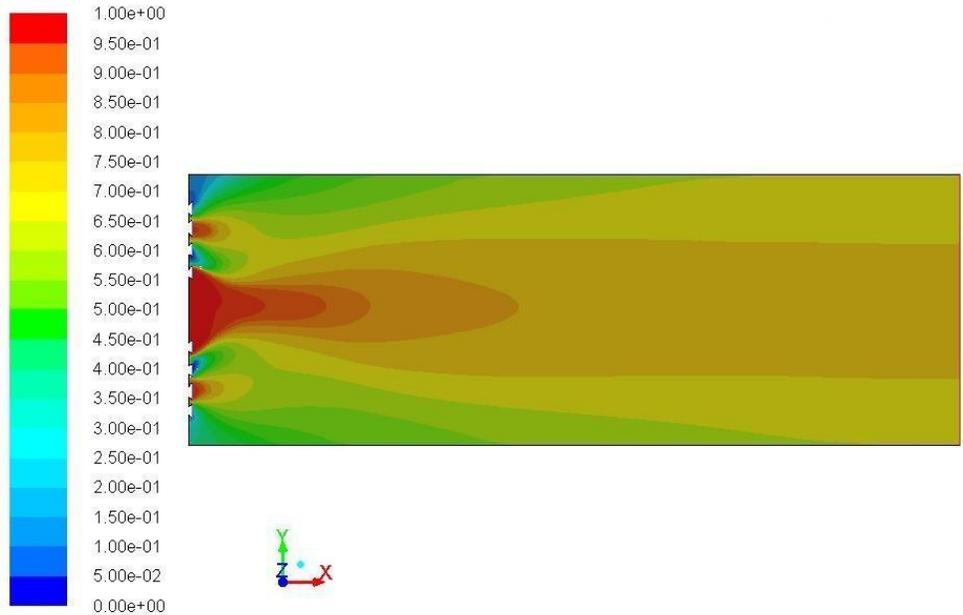
Figure.5.8 Mean Mixture Fraction contour for 45°injection
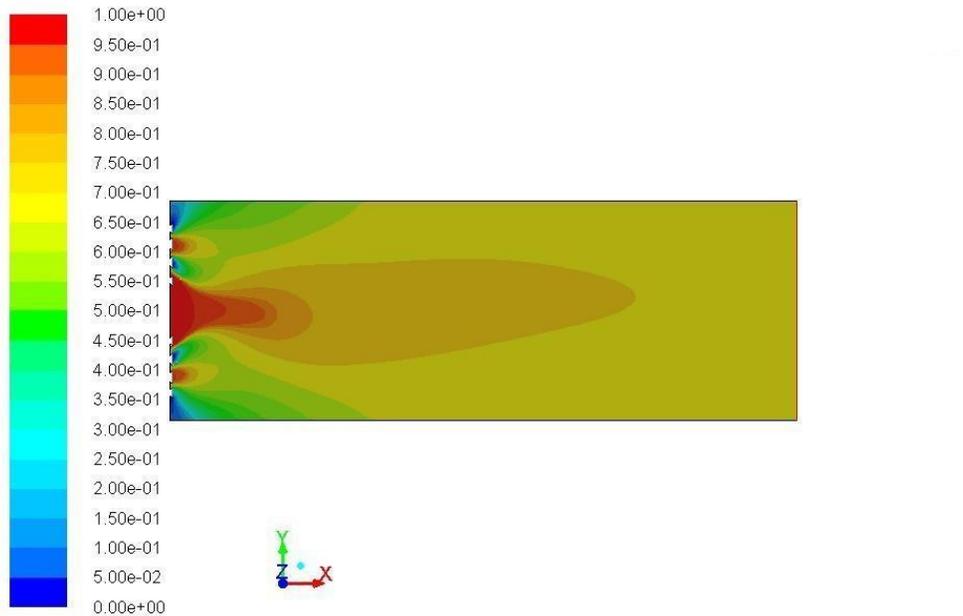
Figure5..9 Mean Mixture Fraction contour for 47.5°injection

Figure.5.10 Mean Mixture Fraction contour for 50°injection

5.3. Temperature Distribution

Figure.5.11 Temperature contour for 40°injection
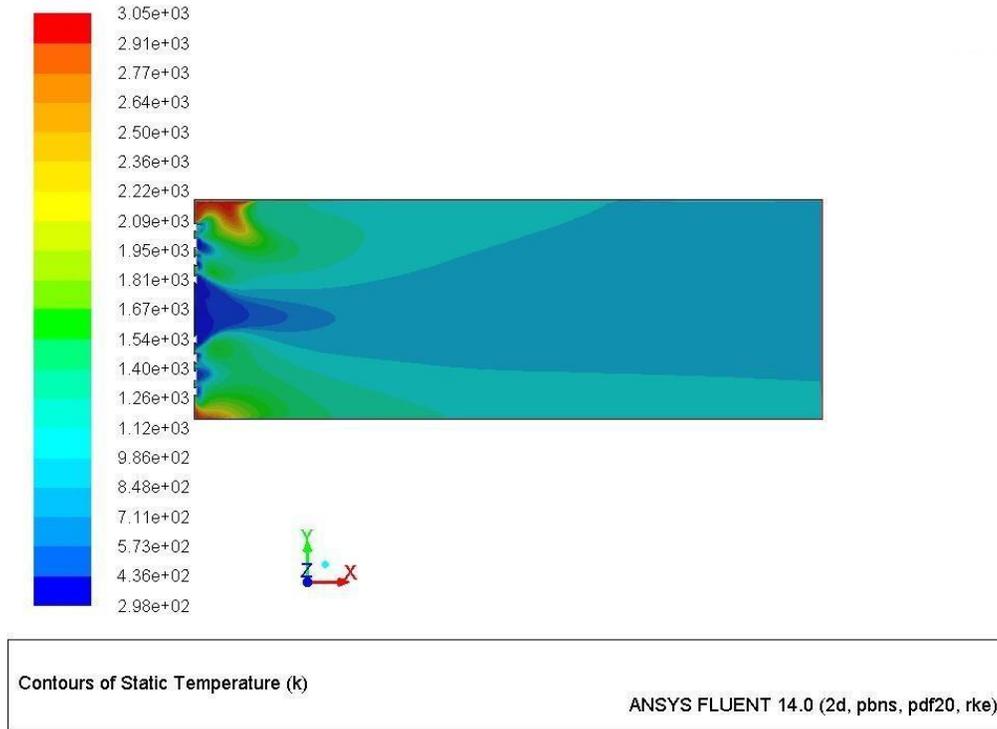
Figure.5.12 Temperature contour for 42.5°injection
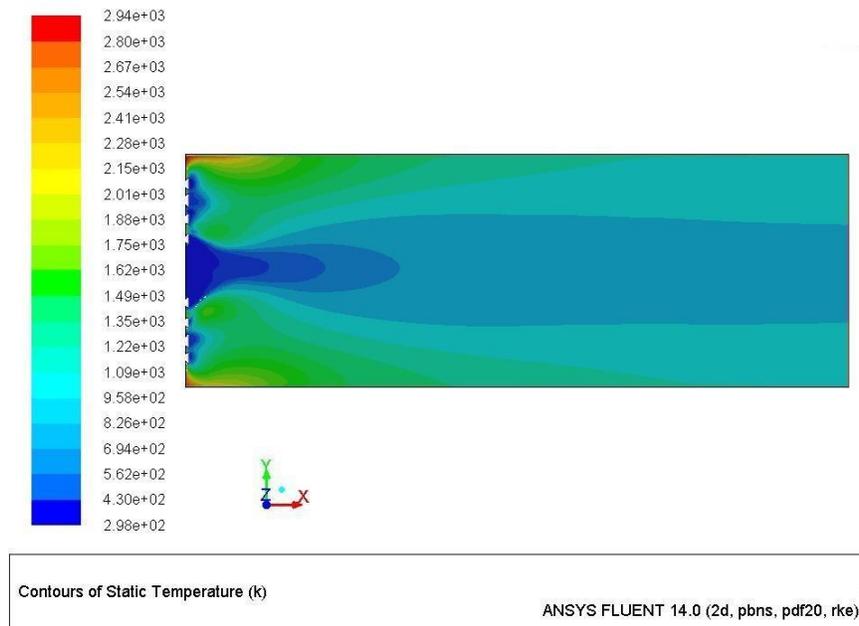

Figure.5.13 Temperature contour for 45°injection
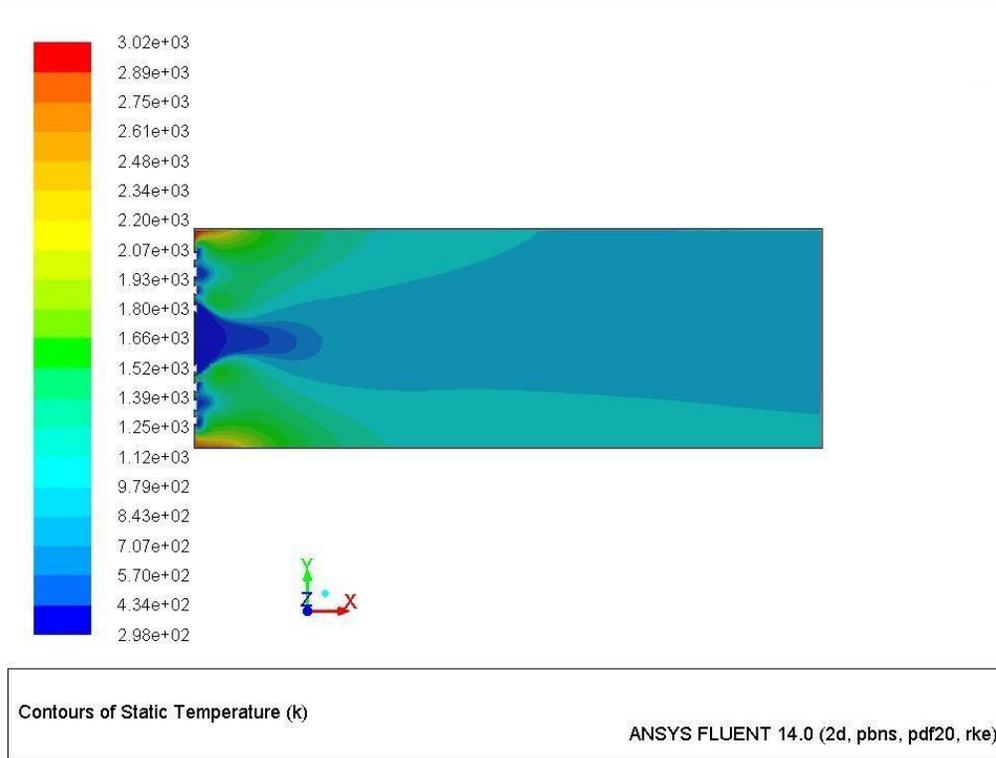
Figure.5.14 Temperature contour for 47.5°injection
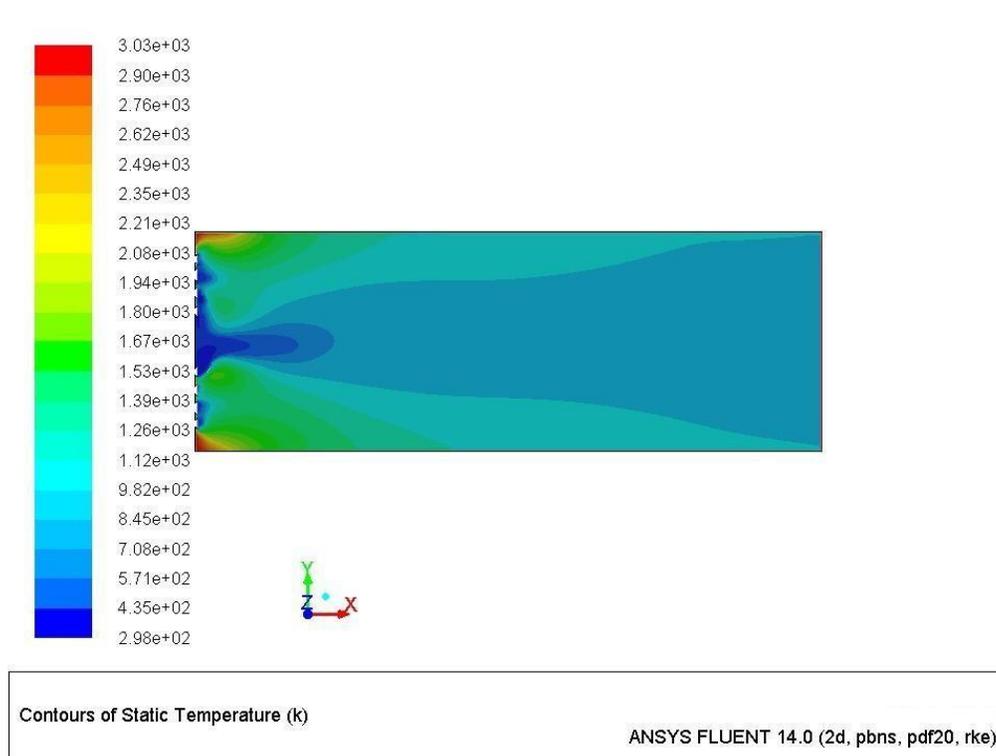
Figure.5.15 Temperature contour for 50°injection

The Temperature distributions in Fuel injector for various angles are given from figure 11 to 15. In figure 11, we have the Temperature of 824^0 K at angle of 40^0. As well as, in figure 12, we have the Temperature of 711^0 K at angle of 42.5^0. In figure 13, we have the Temperature of 826^0 K at angle of 45^0. In figure 14, we have the Temperature of 843^0 K at angle of 47.5^0. Finally, in figure 15, we have the Temperature of

708^0 K at angle of 50^0. By comparing all Temperatures with respect to various angles of attacks, we have

minimum Temp. of 708^0 K at angle of 50^0

5.4. Pressure Distribution

The Pressure distributions in Fuel injector for various angles are given from figure 16 to 20. In figure 16, we have the Pressure of 1.13 x 10^5 Pa at angle of 40^0. As well as, in figure 17, we have the Pressure of 1.1 x 10^5 Pa at angle of 42.5^0. In figure 18, we have the Pressure of 9.7 x 10^4 Pa at angle of 45^0. In figure 19, we have the Pressure of 8.89 x 10^4 Pa at angle of 47.5^0. Finally, in figure 20, we have the Pressure of 8.65 x 10^4 Pa at angle of 50^0. By comparing all velocities with respect to various angles of attacks, we have minimum Pressure of 8.65 x 10^4 Pa at angle of 50^0

Contours of Static Pressure (pascal)

ANSYS FLUENT 14.0 (2d, pbns, pdf20, rke)
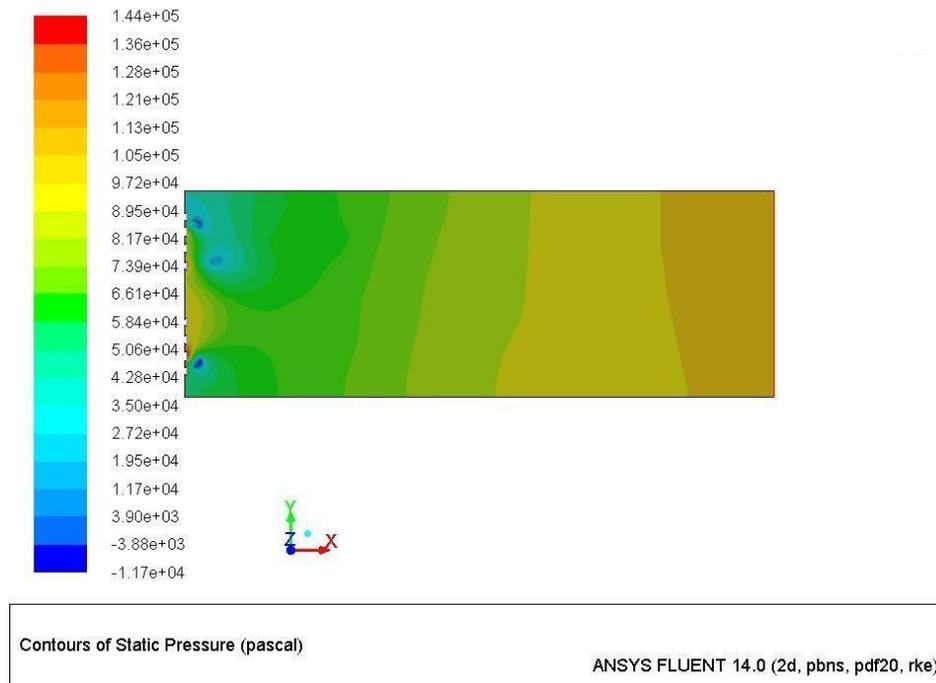
Figure5.16 Pressure contour for 40°injection
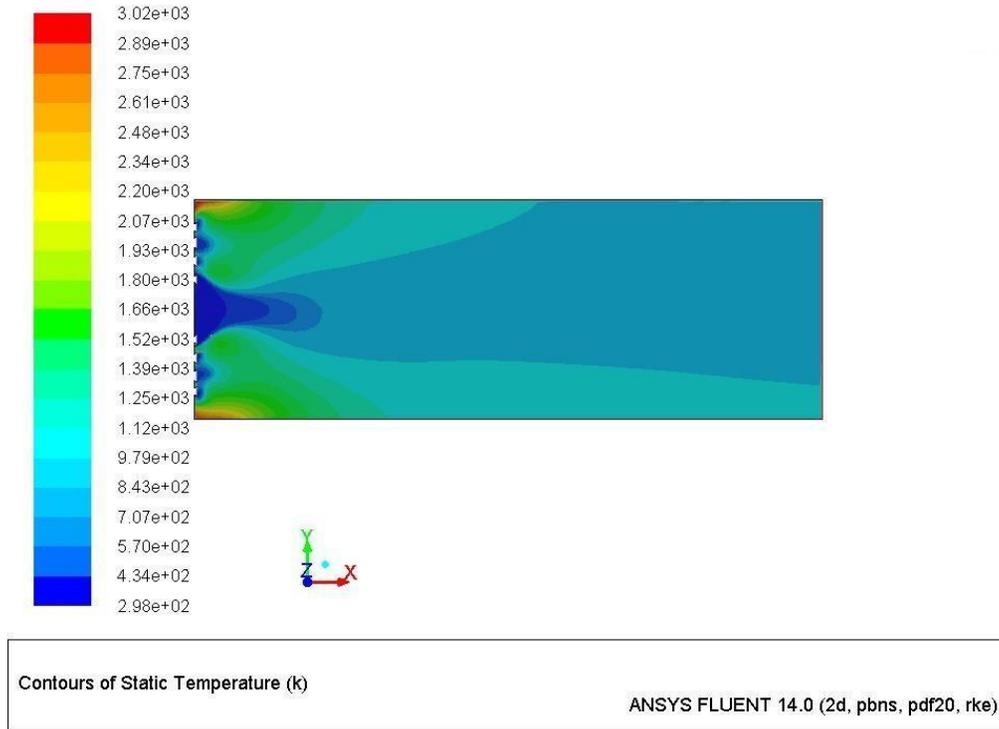
Figure.5.17 Temperature contour for 47.5°injection
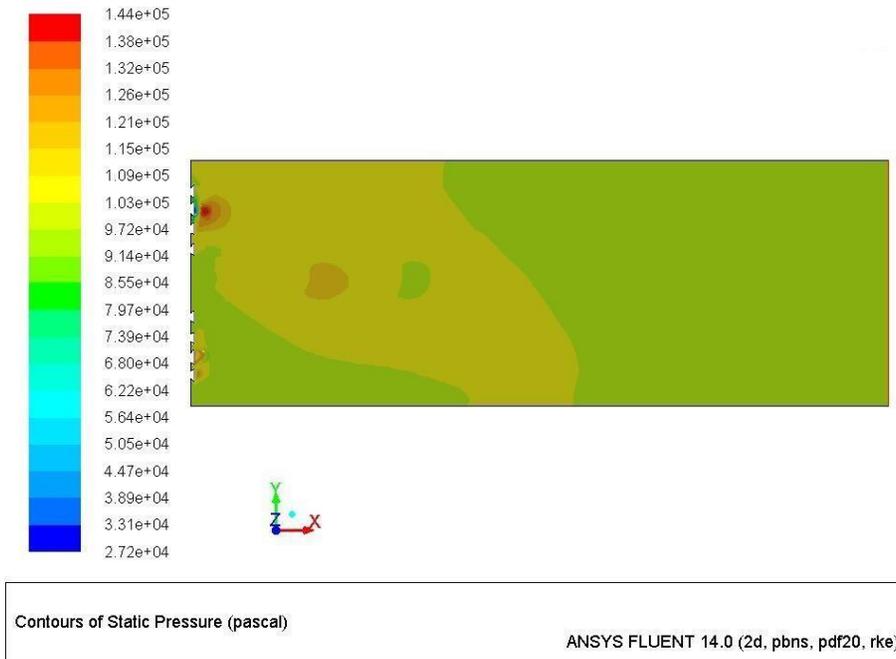
Figure.5.18 Pressure contour for 45°injection
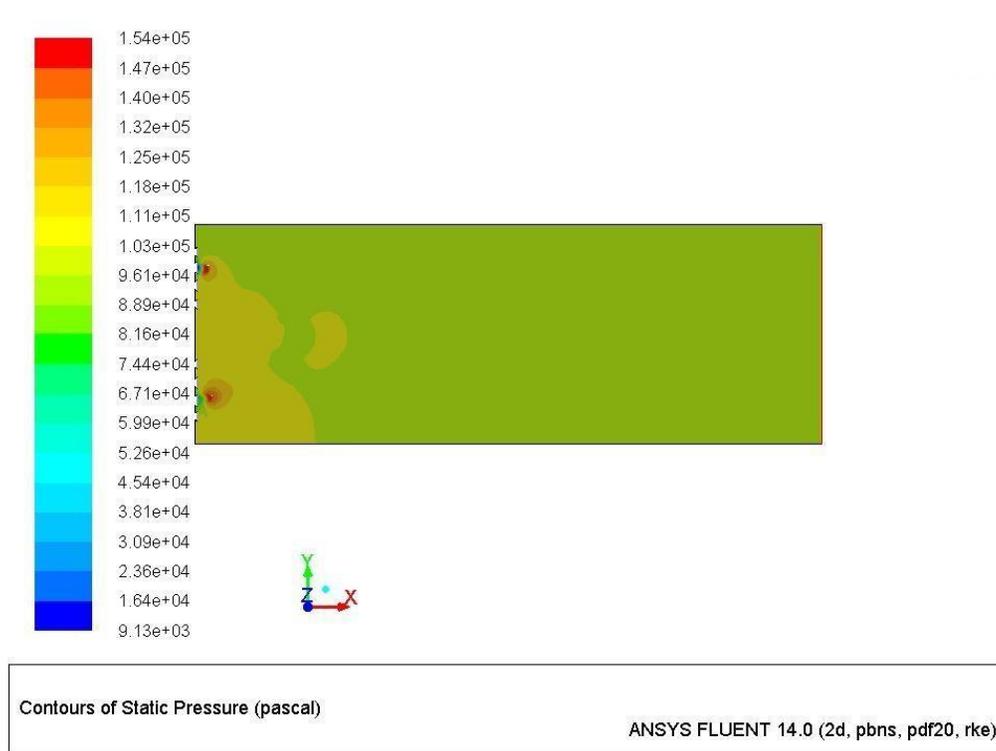
Figure.5.19 Pressure contour for 47.5°injection
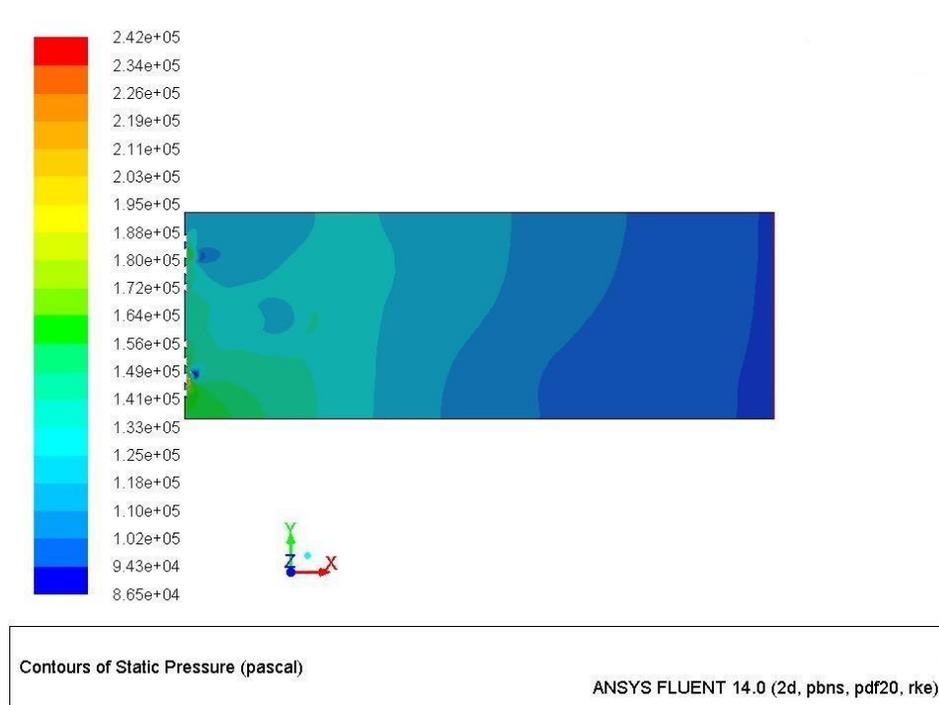
Figure5.20 Pressure contour for 50°injection

6.CONCLUSION

In conclusion, the design, analysis, and simulation of the rocket propulsion system is useful in conducting trade studies of rocket propulsion systems. Chamber dimensions, propellant selections, and injector selection between doublet or triplet had been selected based on the literature survey for the desired rocket system design. A 2-D graphical representation of the injector and/or chamber is plotted and coordinates of the plotted line are displayed. The post- processing simulation of the rocket propulsion system is done in a computational fluid dynamics (CFD) program such as ANSYS ICEM CFD mesh generation software and ANSYS FLUENT. Using ANSYS FLUENT CFD software, the injector is simulated for the flow, turbulence, heat transfer, combustion in the chamber, or various other options of the rocket propulsion system. Based on the simulation and the results observed by the numerical simulation the results are interpreted. Based on the table mentioned in the above chapter, it's clear that the 40° injector design has low velocity with high mean mixture fraction and increase in temperature which is due to the proper combustion in fuel and oxidizer mixture. Further to this the case can be simulated to various altitudes in future in order to determine its performance during all maneuvers and altitudes.

REFERENCES

[1] M Schulze, L.Bock, T.Sattelmayer, W.Polifke, Arbeitsunteriagen zu den Vorlesungen Warmetransportphanomene and Stoffubertngung, Technische Universtiat Munchen, 2013.

[2] O.Bozic, Skript zur Vorlesung Raumfahtantriebe, Technische Universtiat Braunschweig, 2011.

[3] B.M Schneider, Experimentelite Untersunchungen zur Spraystrukfur in transienten, vendampfenden und nicht verdamfenden Bennstoffstrahlen untrt Hochdruck, Eidgenossiche technische Hochschule Zurich, 2003.

[4] R. Schmucker, Hybridraketenantriebe – EineEinfuhrung in theortische und technische Probleme, Goldmann Verlag, 1972

[5] D.K.Huzel, D.H.Huang, H.Arbit, Modern Engineering for Design of Liquid –Propellant Rocket Engines, American Institute of Aeronautics and Astronautics, Progress in Astronautics and Aeronautics, 1992.

[6] E.Gamper, R. Hink, Germany Design and test of Nitrous oxide Injectors for a Hybrid Rocket Engine.

[7] S. Mejía and F. Rojas, 2015, "Cohete AINKAA VI Misión SÉNECA IV. Proyecto deGrado.," Universidad de los Andes.

[8] L. Longas and F. Rojas, 2012, "Puesta a punto y lanzamiento de la misión colombiana de cohetería experimental con propulsión líquida. Proyecto PUA, Misión Seneca VI.," Universidad de los Andes.

[9] J. Cabas and F. Rojas, 2015, "Proyecto PUA: Misión de cohetería experimental SÉNECA VI," Universidad de los Andes.