

Declaration by the Students

We declare that this dissertation represents our ideas in our own words and where others' ideas or words have been included, we have adequately cited and referenced the original sources.

We also declare that we have adhered to all principles of academic honesty and integrity and have not misrepresented or fabricated or falsified any idea/data/fact/source in this dissertation.

We understand that any violation of the above will be cause for disciplinary action by the Institute and can also evoke penal action from the sources which have thus not been properly cited or from whom proper permission has not been taken when needed.

Satyam Ashokrao Avhad
131020004

Sakina Moiz Tinwala
131021039

Date:

Place:

Certificate

This is to certify that **Mr Satyam Ashokrao Avhad and Ms Sakina Moiz Tinwala**, have completed this dissertation, in partial fulfilment of the requirement for the Degree of Bachelor of Technology in Mechanical Engineering, titled **CFD Simulation of Combustion in a Turbojet Engine** to our satisfaction.

Dr. A. V. Deshpande,

Assistant Professor,
Mechanical Engineering Department,
V.J.T.I., Mumbai.

Dr N. P. Gulhane,

Associate Professor and Head,
Mechanical Engineering Department,
V.J.T.I., Mumbai.

Dr O.G. Kakde

Director,
V.J.T.I., Mumbai.

Certificate

The dissertation titled **CFD Simulation Of Combustion In A Turbojet Engine**, submitted by **Mr Satyam Ashokrao Avhad (131020004)** and **Ms Sakina Moiz Tinwala(131021039)**, is found to be satisfactory and is approved for the Degree of Bachelor of Technology in Mechanical Engineering.

(Name & Signature)

Supervisor / Guide

Date:

(Name & Signature)

Examiner

Place:

Abstract

A Numerical investigation of combustion in a miniature turbojet engine was conducted. The experimental setup is located at Automobile Engineering lab of Veermata Jijabai Technological Institute (V. J. T. I.), Mumbai. For the basis to Numerical investigation, reference values were taken from the experiment conducted on Jet engine by **Mr. Pramod Pawar and Mr. Flevian Gonsalves** in the year 2011-12 at Veermata Jijabai Technological Institute (V. J. T. I.), Mumbai. The k- ξ viscous model and non-premixed combustion species model in ANSYS Fluent were used to simulate the fluid flow in the combustion chamber. The dependence of temperature at the exit of the combustion chamber was analysed using this ANSYS model. Improvements in the temperature range was observed by changing input conditions to the turbojet.

Table of Contents

Declaration by the Students	1
Certificate.....	3
Certificate.....	5
Abstract.....	6
List of Figures.....	10
List of Tables	11
1. Introduction.....	12
2. Literature review	13
2.1 Turbojet Engine	13
2.1.2 Compressor.....	14
2.1.2 Combustion chamber.....	15
2.1.3 Combustor liner	16
2.1.4 Turbine	17
2.2 Combustion Process.....	23
2.2.1 Introduction	23
2.2.2 Stages of combustion.....	23
2.2.2.1 Primary	23

2.2.2.2 Secondary	23
2.2.2.3 Tertiary	23
2.2.3 Types of combustion	24
2.2.4 Adiabatic flame temperature	27
2.2.5 Rich flammability limit.....	27
2.2.6 Jet fuel and its kinetics	28
3. Software Details.....	30
3.1 ANSYS Design Modular (DM)	30
3.2 ANSYS Meshing (AM).....	31
3.2.1 Mesh types.....	31
3.2.2 Mesh quality	31
3.3 ANSYS Fluent	34
3.3.1 Models	34
3.3.2 User defined functions (UDF).....	38
3.3.3 Solver.....	39
3.3.3.1 Types.....	39
3.3.3.2 Velocity formulation	40
3.3.3.3 Time	40
3.3.3.3 2D Space	40

4. Experimental Setup	41
4.1 Components	41
4.2 Measurement Instruments	44
4.3 Experimental Results	45
5. Calculations	46
6. Numerical Procedure.....	53
7. Results	60
8. Conclusions	66
9. Future Scope	67
10. References.....	68
11. Acknowledgements	69

List of Figures

Fig 2.1 Turbojet Engine	13
Fig 2.2 The Brayton Cycle	17
Fig 2.3 Non Premixed Combustion	24
Fig 2.4 Partially Premixed Combustion	25
Fig 2.5 Premixed Combustion.....	26
Fig 3.1 Cell Types.....	31
Fig 4.1 Solidworks Model of Flame Tube.....	43
Fig 4.2 Solidworks Model of Complete Combustion Chamber Assembly.....	43
Fig 4.3 Experimental Setup Turbojet Engine	44
Fig 6.1 2D Mesh Model.....	53
Fig 6.2 Injection Properties.....	56
Fig 7.1 Contour of mass fraction of kerosene	60
Fig 7.2 Temperature profile at Spark ignition	61
Fig 7.3 Contour of static temperature	61
Fig 7.4 Contour of total temperature.....	62
Fig 7.5 Contour of Velocity magnitude.....	63
Fig 7.6 Contour of Pressure	64
Fig 7.7 PDF Table	65

List of Table

Table 3.1 Recommended Orthogonal Quality and Skewness	34
Table 4.1 Experimentation Results	45

Chapter 1: Introduction

In the past decade, the use of mini turbojet planes for surveillance as well as military purpose has seen a remarkable growth. The challenges faced in designing such a jet engine are the ability to provide a stable and efficient combustion process and difficulties in controlling pollution and smoke. The higher temperature at the end of the combustion chamber as well as unstable combustion may lead to material failure. Thus, techniques such as cooling the chamber using bypass air have been in use. Also recently, a new technology for cooling turbine blade through surface vents is another trending topic for research. In addition, to reduce pollution various methods like Exhaust gas recirculation (EGR), catalytic reduction are in use.

Computational fluid dynamics (CFD) is a branch of physics that deals with the study of the mechanics of fluid: liquid, plasmas and gases and forces acting on them. CFD is based on Navier- Stokes equations that describe how pressure, velocity, density and temperature of a moving fluid are related. It finds applications widely in various fields -Aerospace, biomedical, automotive sectors. The field of CFD has seen tremendous growth in the last decades and its usage is still expanding into many fields.

By using CFD simulations, emissions can be predicted and thus, an optimized design can be used to minimize the impact of harmful emissions due to formation of NO_x , CO , CO_2 , SO_2 on nature.

A well-established CFD model will help in designing the combustion chamber for required conditions. It also helps to determine exhaust gas species and/or pollutants. Thus, the aim of this project is to establish a CFD model for visualizing the flow inside the combustion chamber and iterate the design for most stable operation by varying few parameters like spark plug location, bypass ratio and location of fuel injector. It will provide pressure and temperature profiles of the entire combustion chamber during the combustion process.

Chapter 2: Literature Review

2.1 Turbojet Engine:

A turbojet engine can be broken down into five major subassemblies,

- Inlet duct
- Compressor
- Combustion chamber
- Turbine wheel
- Exhaust outlet

When the engine is running, air is drawn into the compressor where it is compressed and the pressure is increased. It is then ducted to the combustion chamber where fuel is added and burnt. The heat in the combustion chamber causes the air to expand before it exits through a nozzle and drives the power turbine. The turbine drives the compressor by means of a drive shaft. The remaining hot gases are expelled out through the exhaust nozzle. Forcing large amount of air in the engine and expelling it out through the exhaust duct at a much higher velocity creates thrust. This is the principle of operation of the turbojet engine.

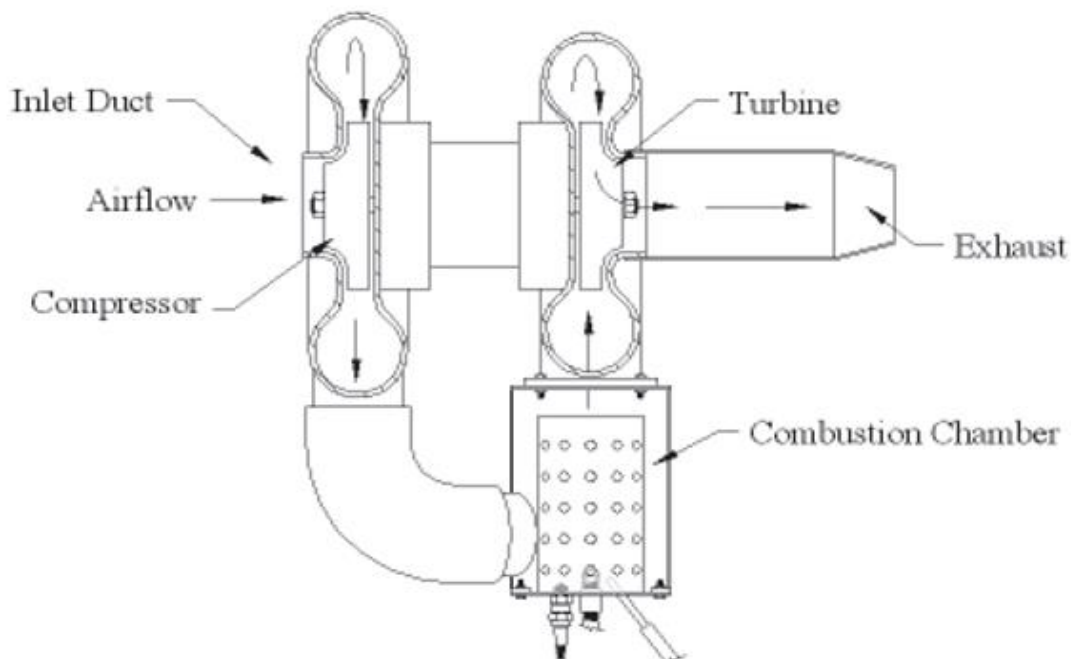


Figure 2.1 Turbojet Engine

2.1.1 Compressor:

A gas compressor is a mechanical device that increases the pressure of a gas by reducing its volume. An air compressor is a specific type of gas compressor.

There are 3 basic types of air compressors

- Reciprocating
- Rotary screw
- Rotary centrifugal

Reciprocating air compressor:

It is a positive displacement machine, and it achieves an increase in pressure using a piston cylinder assembly. It can be single acting or double acting, but it is bulky, has intermittent supply, leakage losses and has a pressure ratio limited to 2-3 per stage therefore it is inefficient for aviation applications.

Rotary Screw:

It is a positive displacement compressor, which uses the motion of two intermeshing lobes for compression and increase in pressure. There are no valves and this type of compressor can provide a smooth pulse free air output.

Centrifugal compressor:

It is a dynamic compressor which depends on transfer of energy from a rotating impeller to the air. The compressor wheel turns at a very high speed usually between 45,000 and 125,000 rpm. It is usually made from an aluminum alloy. The temperature of the air will increase 200 to 400 deg. F in the compressor. The compressed air exits the compressor into a diffuser which is usually a casting that increases in area so that the air will be slowed down and the pressure will increase. The compressor end contains the impeller. The curved portions of the blades near the centre are called the inducer vanes and are used to draw air into the compressor where the radial blades accelerate it. The air then passes into the snail shaped housing called the diffuser.

2.1.2 Combustion Chamber

The combustion chamber is the key element of the engine. This is where fuel is mixed with compressed air and burned, causing the air to expand and drive the turbine wheel. A shield called a “combustion liner” is designed to allow some air to mix with the fuel and burn, while the remainder of the air is used to cool the steel parts.

The combustion chamber must operate over a wide range of conditions. It must withstand high rates of burning, have a minimum pressure drop, be light in weight, and have minimum bulk.

The turbojet is a continuous flow system, therefore, the combustion chamber in a turbojet engine differs from the combustion in a diesel engine.

One of the vital problem associated with the design of turbojet combustion system is to secure a steady and stable flame inside the combustion chamber.

A few difference between the combustion chamber in a diesel engine and a turbojet engine are:

- Combustion in a turbojet engine takes place in a continuous flow system and, therefore the advantage of high pressure and restricted volume available in a diesel engine is lost. The chemical reaction takes place at a relatively slowly thus requiring large residence time in the combustion chamber in order to achieve complete combustion.
- The turbojet engine requires about 100:1 air-fuel ratio by weight, but the air-fuel ratio required for the combustion in diesel engine is approximately 15:1. Therefore, it is impossible to ignite and maintain a continuous combustion with such weak mixture.
- A pilot or recirculated zone should be created in the main flow to establish a stable flame which helps to ignite the combustible mixture continuously.
- A stable continuous flame can be maintained inside the combustion chamber when the stream velocity and fuel burning velocity are equal. Unfortunately, most of the fuels have low burning velocities of the order of a few meters per second, therefore, flame stabilization is not possible unless some technique is employed to anchor the flame in the combustion chamber.

The rate of vapourization of fuel in the combustion chamber depends on the following criteria:

- Surface area of fuel
- Temperature of fuel and chamber
- Pressure
- Rate of flow of air

2.1.3 Combustion Liner:

The combustion liner is a perforated steel liner that fits into the combustion chamber of the turbojet. Fuel is sprayed into this liner and both combustion air and cooling air flows through the combustion liner.

The holes in the combustion liner are adjusted to allow the right amount of air to mix with the fuel so that combustion can occur. If the holes are too large, the incoming pressurized air will blow out the flame. If the holes are too small, there will not be enough oxygen to support combustion. If the holes at the fuel inlet end are too small, the flame will have to travel down the combustion liner until enough oxygen has entered to support combustion. This will cause the combustion to occur in the inlet to the turbine and overheat the turbine.

The normal definition for the bypass ratio (BPR) of a turbojet engine is the ratio of the mass flow rate of the bypass stream to the mass flow rate entering the core.

Therefore holes in a combustion chamber liner are very critical to determine an efficient bypass ratio so that a rich mixture in the primary zone is maintained for ignition of the fuel, effective combustion at stoichiometric ratio in the secondary zone and proper dilution in the tertiary zone is achieved to reduce the concentration of the harmful emissions which are exhausted to the atmosphere.

2.1.4 Turbine:

A turbine is a rotary mechanical device, which consists of a turbine wheel and a turbine housing, converts the engine exhaust gas into mechanical energy to drive the compressor. The gas, which is restricted by the turbine's flow cross sectional area, results in a pressure and temperature drop between the inlet and outlet.

The turbine blades are under thermal stresses, creep and fatigue due to cyclic loading and high temperatures due to exhaust gases, thus the exhaust gas temperatures must be regulated so as to avoid mechanical failure of the turbine.

Therefore, turbines are manufactured with corrosion resistant materials such as Nickel alloys which can withstand such high temperatures from the combustion chamber.

The basic requirements of the turbine are light weight, high efficiency, reliability in operation and long working life. More stages of the turbine are always preferred because it helps to reduce the stresses in the blades and increases the overall life of the turbine.

The cooling of the turbine blades is essential for long life as it is continuously subjected to high temperature gases. There are different methods of cooling the blades, the most common being aircooling in which the air is passed through the holes provided through the blades.

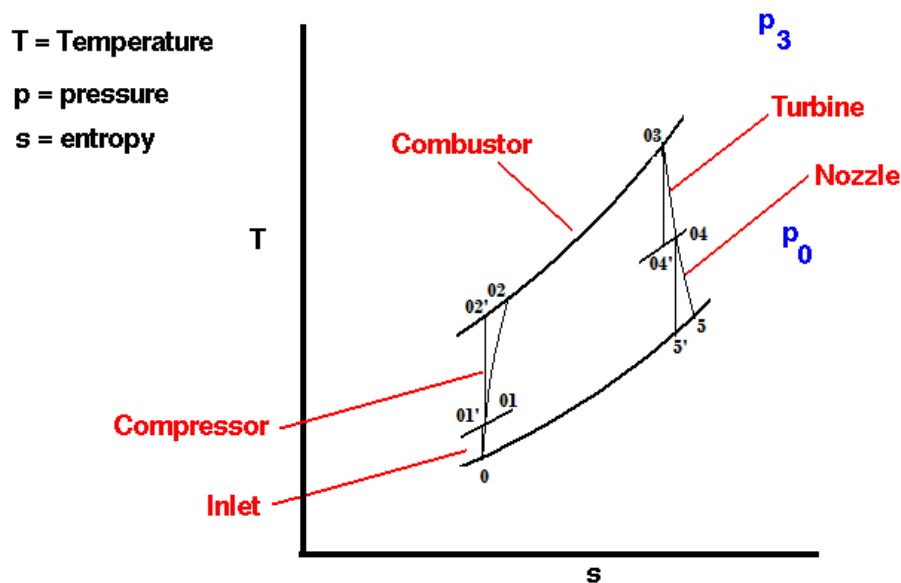


Fig 2.2: Brayton cycle

Various processes occurring in the turbojet engine are depicted in the figure above. The energy equations for these processes are written below.

Inlet diffuser:

Ambient air enters the diffuser at the pressure P_0 , temperature T_0 and velocity C_i . The pressure rises to P_1 at the diffuser exit where the velocity is reduced to C_j . There is no energy transfer; only energy transformation takes place here.

$$h_0 = h_{01}$$

$$\frac{T_{01'}}{T_0} = \left(\frac{P_{01'}}{P_0} \right)^{\frac{\gamma-1}{\gamma}}$$

Therefore pressure ratio,

$$\therefore \frac{P_{01'}}{P_0} = \left(\frac{T_{01'}}{T_0} \right)^{\frac{\gamma}{\gamma-1}}$$

Diffuser efficiency is given by;

$$\eta_d = \frac{P_{01} - P_0}{P_{01'} - P_0}$$

Therefore, the static pressure rise in the diffuser is given by;

$$\frac{P_{01}}{P_0} = \left(1 + \eta_d \frac{\gamma-1}{2} M^2 \right)^{\frac{\gamma}{\gamma-1}}$$

Compressor:

Air enters the compressor at reduced velocity and Mach number. Its pressure and enthalpy is raised to P_{02} and h_{02} in the actual process (01-02). Both energy transformation and transfer occur in this process.

The actual work done by the compressor on the air is given by;

$$W_c = h_{02} - h_{01} = C_p (T_{02} - T_{01})$$

Compressor pressure rise is given by;

$$\frac{P_{02'}}{P_{01}} = \left(\frac{T_{02'}}{T_{01}} \right)^{\frac{\gamma}{\gamma-1}}$$

Now, the process of compression is not an ideal process so the performance of the compressor is given by compressor efficiency;

$$\eta_c = \frac{h_{02'} - h_{01}}{h_{02} - h_{01}}$$

Thus,

$$h_{02} - h_{01} = \frac{1}{\eta_c} (h_{02'} - h_{01}) = \frac{C_p}{\eta_c} (T_{02'} - T_{01})$$

$$W_c = h_{02} - h_{01} = C_p (T_{02} - T_{01})$$

$$= \frac{C_p T_{01}}{\eta_c} \left(\frac{T_{02'}}{T_{01}} - 1 \right) = \frac{C_p T_{01}}{\eta_c} \left[\left(\frac{P_{02}}{P_{01}} \right)^{\frac{\gamma}{\gamma-1}} - 1 \right]$$

Combustion Chamber:

Air enters the combustion chamber from the compressor at pressure P_2 , temperature T_2 . The combustion of the fuel increases the enthalpy of the air-fuel mixture. The mass of high temperature gases flowing from the combustion chamber to the turbine and the propelling nozzle is given by:

$$\dot{m}_j = \dot{m}_i + \dot{m}_f$$

$$\dot{m}_j = \dot{m}_i(1 + f)$$

$$f = \text{fuel_ratio} = \frac{\dot{m}_f}{\dot{m}_i}$$

Heat supplied in the combustion process is given by;

$$q = h_{03} - h_{02} \\ = C_p (T_{03} - T_{02})$$

Again due to incomplete combustion the combustion process is inefficient the performance of the combustion chamber is given by;

$$\eta_{comb} = \frac{h_{03} - h_{02}}{f \cdot Q_f}$$

Where Q_f is the specific heat of the fuel.

Turbine:

The products of the combustion from the combustion chamber enter the turbine at the pressure P_3 , temperature T_3 . Properties of these gases are different from those of the air flowing through the inlet diffuser and compressor. Expansion of the gases through the turbine and the nozzle is shown in the cycle. Both the energy transformation and transfer occur in this process.

Work done by turbine is given by,

$$W_T = h_{03} - h_{04} = C_p (T_{03} - T_{04})$$

Pressure ratio of the turbine is given by;

$$\frac{P_{03}}{P_{04}} = \left(\frac{T_{03}}{T_{04}} \right)^{\frac{\gamma}{\gamma-1}}$$

Turbine efficiency is given by;

$$\eta_T = \frac{T_{03} - T_{04}}{T_{03} - T_{04}'}$$

Therefore;

Turbine Work =

$$h_{03} - h_{04} = \eta_T C_p T_{03} \left[1 - \left(\frac{P_{04}}{P_{03}} \right)^{\frac{\gamma}{\gamma-1}} \right]$$

Also work done by the turbine is used to drive the compressor therefore;

$$h_{02} - h_{01} = h_{03} - h_{04}$$

$$\frac{C_p T_{01}}{\eta_c} \left[\left(\frac{P_{02}}{P_{01}} \right)^{\frac{\gamma}{\gamma-1}} - 1 \right] = \eta_T C_p T_{03} \left[1 - \left(\frac{P_{04}}{P_{03}} \right)^{\frac{\gamma}{\gamma-1}} \right]$$

i.e.

Nozzle:

Exhaust gases from the turbine enter the propelling nozzle at pressure P_4 , temperature T_4 . The gases expand adiabatically to the exit pressure P_5 . There is no energy transfer; only energy transformation occurs here.

$$h_{04} = h_{05},$$

Nozzle pressure ratio is given by;

$$\frac{P_{04}}{P_{05}} = \left(\frac{T_{04}}{T_{05}} \right)^{\frac{\gamma}{\gamma-1}}$$

Nozzle efficiency is defined as the ratio of the actual and isentropic values of the enthalpy drop.

$$\eta_{noz} = \frac{h_{04} - h_5}{h_{04} - h_{5s}}$$

Now change in enthalpy in nozzle is given by;

$$h_{04} - h_5 = \eta_{noz} C_p T_{04} \left[1 - \left(\frac{P_5}{P_{04}} \right)^{\frac{\gamma}{\gamma-1}} \right]$$

Also;

$$h_{04} - h_5 = \frac{C_j^2}{2}$$

$$\therefore \frac{C_j^2}{2} = \eta_{noz} C_p T_{04} \left[1 - \left(\frac{P_5}{P_{04}} \right)^{\frac{\gamma}{\gamma-1}} \right] \quad \text{or} \quad C_j = \sqrt{2 \eta_{noz} C_p T_{04} \left[1 - \left(\frac{P_5}{P_{04}} \right)^{\frac{\gamma}{\gamma-1}} \right]}$$

Thrust:

The force which propels the object forward is called as thrust or propulsive force. In case of the turbojet engine this propulsive force is developed by the jet at the exit of the propelling nozzle. Propulsive force developed by the nozzle is given by;

$$\therefore \text{Thrust} = \dot{m}_i [(1 + f) c_j - c_i]$$

The thrust per Kg of the air flow is known as Specific thrust and is given by;

$$I_{SP} = \frac{F}{\dot{m}_a} = (c_j - c_i)$$

Now; The specific static thrust is given by;

$$c_i = 0 \quad (I_{SP})_{St} = c_j \quad (I_{SP})_{St} = \text{specific_static_thrust}$$

Hence, to develop a large specific thrust jet speed should be as large as possible.

Let Δh_{\max} be the enthalpy drop in the nozzle corresponding to the velocity C_j , then jet speed is given by;

$$\frac{C_j^2}{2} = \Delta h_{\max}$$

$$C_j = \sqrt{2 C_p \Delta T}$$

2.2 Combustion Process:

2.2.1 Introduction:

Combustion process includes different burning zones called primary, intermediate and dilution zone. Primary burning take place right after fuel injection with swirling air that helps injection of fuel. In the primary zone burning is also called rich burning where fuel is in excess amount than the air supplied to oxidize the fuel completely. Intermediate zone completes the combustion process where extra amount of air comes through some holes from the annular space. Finally, dilution air is added before it goes to the turbine through dilution zone.

2.2.2 Stages of combustion:

2.2.2.1 Primary zone:

Nearly, 15 to 20% of the total air is passed around the jet of fuel providing a rich mixture in the primary zone. The mixture burns continuously in the primary (pilot) zone and produces high temperature gases.

2.2.2.2 Intermediate zone:

It is also known as secondary zone, about 30% of the total air is supplied in the secondary zone through the annulars around the flame tube to complete the combustion. The secondary air must be admitted at right points in the combustion chamber otherwise the cold injected air may chill the flame locally thereby reducing the rate of reaction. The secondary air helps to complete the combustion as well as helps to cool the flame tube.

2.2.2.3 Dilution zone:

It is also known as tertiary zone, the remaining 50% air is mixed with the burnt gases in the tertiary zone to cool the gases down to the temperature suited to the turbine blade materials.

Sufficient turbulence must be created in all three zones of combustion and uniform mixing of hot and cold gases to give uniform temperature gas stream at the outlet of the combustion chamber.

2.2.3 Types of combustion:

2.2.3.1 Non-Premixed Combustion:

In non-premixed combustion, fuel and oxidizer enter the reaction zone in distinct streams. Non-premixed modeling involves the solution of transport equations for one or two conserved scalars (the mixture fractions). Equations for individual species are not solved. Instead, species concentrations are derived from the predicted mixture fraction fields. Interaction of turbulence and chemistry is accounted for with an assumed-shape Probability Density Function (PDF).

The non-premixed modeling approach has been specifically developed for the simulation of turbulent diffusion flames with fast chemistry. For such systems, the method offers many benefits over the eddy-dissipation formulation. The non-premixed model allows intermediate (radical) species prediction, dissociation effects, and rigorous turbulence-chemistry coupling. The method is computationally efficient in that it does not require the solution of a large number of species transport equations. When the underlying assumptions are valid, the non-premixed approach is preferred over the eddy-dissipation formulation.

- Separate streams for Fuel and oxidizer
- Convection or diffusion of reactants from either side into a flame sheet
- Turbulent eddies distort the laminar flame shape and enhance mixing
- May be simplified to a mixing problem

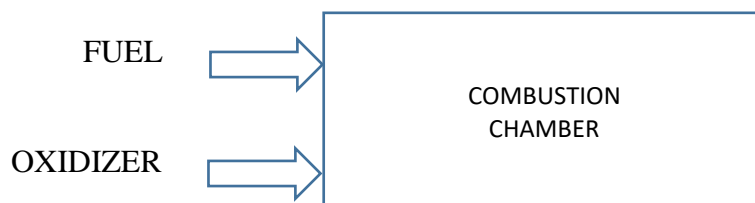


Figure 2.3 Non Premixed Combustion

2.2.3.2 Partially premixed combustion:

Partially premixed combustion systems are premixed flames with non-uniform fuel-oxidizer mixtures (equivalence ratios). Such flames include premixed jets discharging into a quiescent atmosphere, lean premixed combustors with diffusion pilot flames and/or cooling air jets, and imperfectly mixed inlets.

The partially premixed model is a simple combination of the non-premixed model and the premixed model. The premixed reaction-progress variable c , determines the position of the flame front. Behind the flame front ($c=1$), the mixture is burnt and the equilibrium or laminar flamelet mixture fraction solution is used. Ahead of the flame front ($c=0$), the species mass fractions, temperature, and density are calculated from the mixed but unburnt mixture fraction. Within the flame ($0 < c < 1$), a linear combination of the unburnt and burnt mixtures is used.

- Reacting systems with both non- premixed and premixed fuel/oxidizer streams

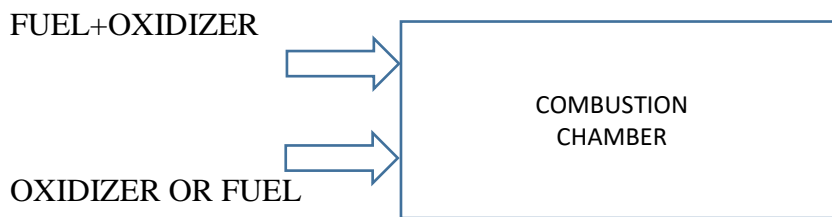


Figure 2.4 Partially Premixed Combustion

2.2.3.3 Premixed combustion:

Premixed turbulent combustion models is based on the reaction-progress variable approach. In premixed combustion, fuel and oxidizer are mixed at the molecular level prior to ignition. Combustion occurs as a flame front propagating into the unburnt reactants. Examples of premixed combustion include aspirated internal combustion engines, lean-premixed gas turbine combustors, and gas-leak explosions.

Premixed combustion is much more difficult to model than non-premixed combustion. The reason for this is that premixed combustion usually occurs as a thin, propagating flame that is stretched and contorted by turbulence. For subsonic flows, the overall rate of propagation of the flame is determined by both the laminar flame speed and the turbulent eddies. The laminar flame speed is determined by the rate that species and heat diffuse upstream into the reactants and burn. To capture the laminar flame speed, the internal flame structure would need to be resolved, as well as the detailed chemical kinetics and molecular diffusion processes. Since practical laminar flame thicknesses are of the order of millimeters or smaller, resolution requirements are usually unaffordable.

The effect of turbulence is to wrinkle and stretch the propagating laminar flame sheet, increasing the sheet area and, in turn, the effective flame speed. The large turbulent eddies tend to wrinkle and corrugate the flame sheet, while the small turbulent eddies, if they are smaller than the laminar flame thickness, may penetrate the flame sheet and modify the laminar flame structure.

In premixed flames, the fuel and oxidizer are intimately mixed before they enter the combustion device. Reaction then takes place in a combustion zone that separates unburnt reactants and burnt combustion products.

- Fuel and oxidizer are already mixed at the molecular level prior to ignition.
- Cold reactants propagate into hot products
- Rate of propagation (flame speed) depends on the internal flame structure
- Much more difficult to model than non-premixed combustion problems
- Turbulence distorts the laminar flame shape and thus accelerates flame propagation

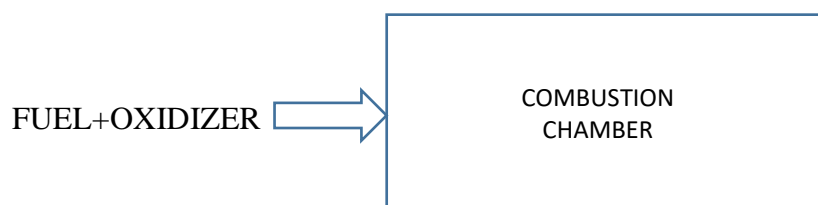


Figure 2.5 Premixed Combustion

2.2.4 Adiabatic Flame Temperature:

For a combustion process that takes place adiabatically with no shaft work, the temperature of the products is referred to as the adiabatic flame temperature [14]. This is the maximum temperature that can be achieved for given reactants. Heat transfer, incomplete combustion, and dissociation all result in lower temperature. The maximum adiabatic flame temperature for a given fuel and oxidizer combination occurs with a stoichiometric mixture (correct proportions such that all fuel and all oxidizer are consumed). The amount of excess air can be tailored as part of the design to control the adiabatic flame temperature.

Adiabatic flame temperature for kerosene is 2394.15 K, under following conditions:

- $P = 2.0685\text{bar}$;
- Initial temperature = 300 K
- Air molar ratio nitrogen / oxygen: 3.76
- Equivalence ratio = 1

2.2.5 Rich Flammability Limit

Rich limit on the mixture fraction can be defined when the equilibrium chemistry option is used. Input of the rich limit is accomplished by specifying a value of the Rich Flammability Limit for the appropriate Fuel Stream, Secondary Stream, or both. The Rich Flammability Limit will not be specified, if the empirical definition option for fuel composition is used.

FLUENT will compute the composition at the rich limit using equilibrium. For mixture fraction values above this limit, FLUENT will suspend the equilibrium chemistry calculation and will compute the composition based on mixing, but not burning, of the fuel with the composition at the rich limit. A value of 1.0 for the rich limit implies that equilibrium calculations will be performed over the full range of mixture fraction. When you use a rich limit that is less than 1.0, equilibrium calculations are suspended whenever f , f_{fuel} or f_{sec} exceeds the limit. This RFL model is a useful approach in hydrocarbon combustion calculations, allowing you to bypass complex equilibrium calculations in the fuel-rich region. The efficiency of RFL will be especially important when your model is non-adiabatic, speeding up the preparation of the look-up tables.

For combustion cases, a value 10% – 50% larger than the stoichiometric mixture fraction can be used for the rich flammability limit of the fuel stream. In this case, the stoichiometric fraction is 0.0563; therefore 0.063 that is around 12% higher is assumed as the rich flammability case for the current model.

2.2.6 Jet fuel and its kinetics:

Jet fuel is a clear to straw-coloured fuel, based on either an unleaded kerosene (Jet A-1), or a naphtha-kerosene blend (Jet B). Jet A, Jet A-1 and Jet B are the most commonly used fuels for commercial aviation. Jet fuel is a mixture of a large number of different hydrocarbons. The range of their sizes (molecular weights or carbon numbers) is defined by the requirements for the product, such as the freezing or smoke point. Kerosene-type jet fuel (including Jet A and Jet A-1) has a carbon number distribution between about 8 and 16 (carbon atoms per molecule); wide-cut or naphtha-type jet fuel (including Jet B), between about 5 and 15.

Jet fuel combustion kinetics is extremely important in order to develop a model that will predict combustion in a jet engine. Although kinetics of jet fuel are still under-developed, significant progress has been made in this area in the recent decades. Development of detailed chemical kinetic models is extremely challenging. Gasoline, diesel and jet fuels derived from different sources are composed of hundreds to thousands of compounds [11]. However, detailed kinetic models for such fuels cannot contain all the compounds due to the limitation of current computational resources. Because of that, a simplified mixture called surrogate mixture must be defined before attempting to develop a kinetic model. Sometimes the fact of limited computational resources can be addressed by reducing detailed kinetics by some optimization techniques. Jet fuels are kerosene-type cut of petroleum containing C-10 to C-18 hydrocarbons, including alkanes, cycloalkanes and aromatic compounds. The criteria and process of developing a surrogate mixture are not unique. However, a proper surrogate fuel must have equivalent physical and chemical properties as the fuel it is representing. Violi et al [12] developed a JP-8 surrogate based on the following criteria:

1. It was assumed that chemical kinetics for each candidate fuel is known.
2. Simplicity must be maintained due to limited computational capabilities.

3. The surrogate is required to match practical fuels in both physical and chemical properties: (a) volatility - boiling range and flash point; (b) sooting tendency - smoking point and luminous number; (c) combustion property - heat of combustion, flammability, and reaction rates. Based on these criteria they developed three surrogate mixtures for JP-8 fuels. The most extensive one has 15% m-Xylene, 10% isooctane, 20% methylcyclohexane, 30% dodecane, 20% tetradecane, 5% tetralin by volume. Humer et al [13] proposed three components surrogate model for jet fuels based on 60 % n-alkanes, 20 % cycloalkanes and 20% aromatics. Aksit and Moss have developed a simple surrogate mixture to reproduce the sooting behavior of aviation kerosene. Their surrogate mixture includes 20% propyl benzene and 80% n-decane by mass. Dagaut et al have developed 1 to 3 component surrogate fuel in order to reproduce kinetics for kerosene combustion. Wang has developed one component ($C_{12}H_{23}$) surrogate fuel based on thermophysical characterization of kerosene combustion. It is also observed that, in addition to separate effort to model a surrogate mixture all the kinetics modeling for a specific named fuel (jet, diesel fuel) starts by defining a surrogate mixture first. Dagaut et al [10] have done an extensive literature survey for the chemical kinetics of combustion of jet fuel. For kinetic model development, general procedures require data such as concentration profile versus time, concentration profile versus temperature, concentration profile versus distance to the burner

Chapter 3: Software Details

With growing computational facilities CFD became a cynosure of many researchers and industrial personnel. Due to their increased demand several commercial CFD solver packages are also available nowadays. ANSYS, Inc. has two different solvers i.e. FLUENT and CFX. Both of the solvers come as integrated packages with geometry modeling, grid generation and post-processing. FLUENT is seen as a more general code while CFX has traditionally been focused on turbo-machinery applications.

3.1 ANSYS Design Modular:

The first step for solving a CFD problem is to define the boundary of fluid volume which is to be simulated. FLUENT provides a feature called Design Modular (DM) which is one the pre-processor of main solver. The first step is to create geometry and after geometry creation the geometry can be transferred to 'Mesh' and geometry can be gridded by that module. After meshing the geometry it is necessary to load the gridded geometry into 'Setup'.

In the 'Setup' user will be able to define boundary conditions and make it ready to send it to the actual fluent solver ('Solution'). Finally solutions can be visualized and other operations can be done using the fluent post-processor ('Results'). The purpose DM is to create a 2D/3D geometry of the fluid. 3D geometries are created from 2D sketches by some operation i.e. 'Extrude', 'Revolve', etc. This DM can also be used to import a dirty CAD geometry and perform various operations to prepare it for meshing. It also allows certain dimension parameterization which is indispensable if user wants to optimize certain geometry. In this work we have used DM to build the fluid geometry.

3.2 ANSYS Meshing:

The second pre-processor of the FLUENT is ANSYS Meshing. The purpose of the ANSYS Meshing is to mesh the fluid volume in an efficient way so that solver can produce a converged results in a comparable less amount of time.

3.2.1 Mesh Type:

There are six meshing method in the ANSYS Meshing for 3-D geometries, which are Tetrahedrons, Sweep, MultiZone, Hex Dominant, Automatic and Cutcell. For 2D geometries it can generate triangular cells, quadrilateral cells or a mixture of quadrilateral and triangular cells.

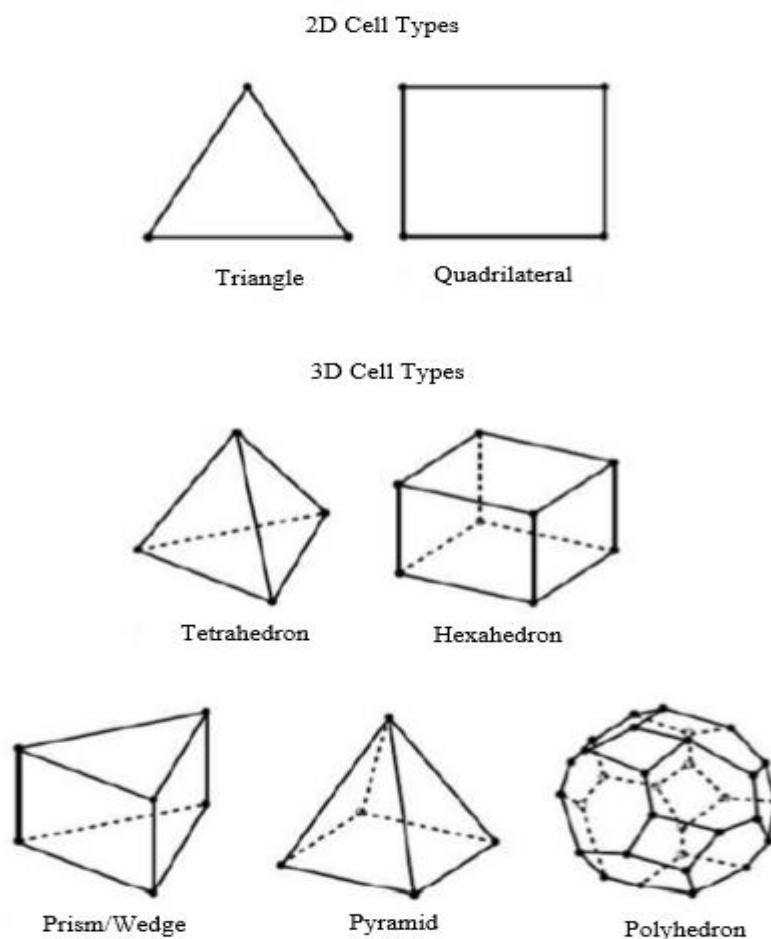


Figure 3.1 Cell Types

As the name suggests Tetrahedrons method produces tetrahedron elements. Sweeps generates prisms or hexahedral elements. MultiZone and Hex Dominant method produce mainly hexahedral with some other types of elements. Automatic method is an integrated method that combines multiple methods based on complexity of the geometry and produces a relatively better mesh, not necessarily all the time. Cutcell method generates cells by cutting the fluid volume directly in a Cartesian coordinate and this method mainly produces hexahedral elements.

Other features of ANSYS Meshing include 'Inflation', 'Match control', 'Global Mesh Control' and different size control tools etc. The purpose of 'inflation' is to add extra elements near wall so that solver can properly capture the aspects of boundary layer. 'Match control' matches the mesh on two faces or two edges on a body element. Matching of mesh on two faces is important for problem involving periodic boundary conditions. As periodic boundaries are supposed to have same solutions, it is necessary to replicate the faces in mesh also. And FLUENT won't allow setting the faces periodic unless the faces match each other. 'Global mesh control' is a graphical user interface (GUI) that allows controlling minimum and maximum cell-size, meshing algorithm and to check mesh-qualities.

Mesh quality control is an important aspect in CFD simulation. ANSYS Meshing provide several mesh quality measurement tools such as orthogonal quality, skewness, maximum corner angle, wrapping factor, Jacobian ratios, aspect ratios etc. However actual mesh quality can be determined from the values of orthogonal quality, skewness metrics.

3.2.2. Mesh Quality:

- **Aspect Ratio:**

It is defined as the ratio of Maximum elemental edge length to the Minimum elemental edge length.

Ideal Value: 1

Acceptable Value: <5

- **Skewness:**

Skewness is one the primary quality measures for a mesh. Skewness determines how close to ideal (equilateral or equiangular) a face or cell is. In the equilateral volume based deviation method, skewness is defined as

$$\text{Skewness} = \frac{\text{optimum cell size} - \text{cell size}}{\text{cell size}}$$

Skewness can be also defined by normalized angle deviation method and mathematically,

$$\text{Skewness} = \max\left[\frac{\theta_{\max} - \theta}{180 - \theta}, \frac{\theta - \theta_{\min}}{\theta}\right]$$

Where,

Θ_{\max} = Largest angle in the face or cell

Θ_{\min} = Smallest angle in the face or cell

θ = Angle for an equiangular face or cell, i.e. 60, 90 degree

In which it suggests how the angle of the element faces are deviated from an ideal equiangular element.

- **Orthogonal quality:**

Orthogonal quality is another major quality measure for a mesh. Orthogonal quality for a cell is computed as minimum of the following quantities computed for each face i :

$$\frac{\bar{A}_i \cdot f_i}{|A_i||f_i|} \text{ and } \frac{\bar{A}_i \cdot c_i}{|A_i||c_i|}$$

Where,

A_i is the face normal vector

f_i is a vector from the centroid of the cell to the centroid of that face

c_i is a vector from the centroid of the cell to the centroid of the adjacent cell that share the face.

Low orthogonal quality or high skewness values are not recommended.

ANSYS, Inc. recommends to consider the following table as general guideline.

Recommendation	Orthogonal Quality	Skewness
Excellent	0.95-1.00	0-0.25
Very Good	0.70-0.95	0.25-0.50
Good	0.20-0.69	0.50-0.80
Acceptable	0.10-0.20	0.80-0.94
Bad	0.001-0.10	0.95-0.97
Inacceptable	0-0.001	0.98-1.00

Table 3.1

3.3 ANSYS Fluent:

ANSYS Fluent is the most-powerful computational fluid dynamics (CFD) software tool available, empowering us to go further and faster to optimize our product's performance. Fluent includes well-validated physical modeling capabilities to deliver fast, accurate results across the widest range of CFD and multi-physics applications.

3.3.1 Models:

The turbulent characteristic is used to improve and accelerate the combustion process inside the engine. The Fluent software uses many turbulence models to solve the turbulent flow issues. These include Spalart-Allmaras model, Transition SST model, Reynolds stress model (RSM), k-omega model, and k-epsilon model.

3.3.1.1 κ - ϵ viscous model:

The κ - ϵ model is the most common model used in CFD codes to simulate the flow with turbulent conditions. This model is applied for fully turbulent flow. It based on two transport equations which are the turbulent kinetic energy (κ) equation and the turbulent dissipation (ϵ) equation. The turbulent length scale is estimated from two properties of the turbulence field, usually the turbulent kinetic energy and its dissipation rate. The dissipation rate of the turbulent kinetic energy is provided from the solution of its transport equation. The κ - ϵ model was proposed by Launder and Spalding. It has many applications in engineering and industry.

3.3.1.1.1 Standard k- ϵ model:

The standard k- ϵ model in FLUENT falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism.

As the strengths and weaknesses of the standard k- ϵ model have become known, improvements have been made to the model to improve its performance. Two of these variants are available in FLUENT: the RNG k- ϵ model and the realizable k- ϵ model.

The standard k- ϵ model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ϵ). The model transport equation for k is derived from the exact equation, while the model transport equation for ϵ was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart.

In the derivation of the k- ϵ model, the assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k- ϵ model is therefore valid only for fully turbulent flows.

3.3.1.1.2 RNG model:

The RNG k- ϵ model was derived using a rigorous statistical technique (called renormalization group theory). It is similar in form to the standard k- ϵ model, but includes the following refinements:

- The RNG model has an additional term in its ϵ equation that significantly improves the accuracy for rapidly strained flows.

- The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows.
- The RNG theory provides an analytical formula for turbulent Prandtl numbers, while the standard k - ϵ model uses user-specified, constant values.
- While the standard k - ϵ model is a high-Reynolds-number model, the RNG theory provides an analytically derived differential formula for effective viscosity that accounts for low-Reynolds-number effects. Effective use of this feature does, however, depend on an appropriate treatment of the near-wall region.

These features make the RNG k - ϵ model more accurate and reliable for a wider class of flows than the standard k - ϵ model.

The RNG-based k - ϵ turbulence model is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called "renormalization group" (RNG) methods . The analytical derivation results in a model with constants different from those in the standard model, and additional terms and functions in the transport equations for k and ϵ .

3.3.1.1.3 Realizable model:

The realizable k - ϵ model is a relatively recent development and differs from the standard k - ϵ model in two important ways:

- The realizable k - ϵ model contains a new formulation for the turbulent viscosity.
- A new transport equation for the dissipation rate ϵ , has been derived from an exact equation for the transport of the mean-square vorticity fluctuation.

The term "realizable" means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Neither the standard k - ϵ model nor the RNG k - ϵ model is realizable.

An immediate benefit of the realizable k - ϵ model is that it more accurately predicts the spreading rate of both planar and round jets. It is also likely to provide superior performance

for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation.

3.3.1.2 P1 Radiation model:

The P-1 radiation model is the simplest case of the more general P-N model, which is based on the expansion of the radiation intensity I into an orthogonal series of spherical harmonics [8]. The P-1 model should typically be used for optical thicknesses >1 [9].

Advantages and Limitations of the P-1 Model

The P-1 model has several advantages, for the P-1 model, the Reynolds transport equation is a diffusion equation, which is easy to solve with little CPU demand. The model includes the effect of scattering. For combustion applications where the optical thickness is large, the P-1 model works reasonably well. In addition, the P-1 model can easily be applied to complicated geometries with curvilinear coordinates.

Limitations when using the P-1 radiation model:

- The P-1 model assumes that all surfaces are diffuse. This means that the reflection of incident radiation at the surface is isotropic with respect to the solid angle.
- The implementation assumes gray radiation.
- There may be a loss of accuracy, depending on the complexity of the geometry, if the optical thickness is small.
- The P-1 model tends to overpredict radiative fluxes from localized heat sources or sinks.

3.3.1.3 Species models:

Non-premixed Combustion model

In non-premixed combustion, fuel and oxidizer enter the reaction zone in distinct streams. This is in contrast to premixed systems, in which reactants are mixed at the molecular level before burning. Examples of non-premixed combustion include pulverized coal furnaces,

diesel internal-combustion engines and pool fires. Under certain assumptions, the thermochemistry can be reduced to a single parameter: the mixture fraction. The mixture fraction, denoted by Y , is the mass fraction that originated from the fuel stream. In other words, it is the local mass fraction of burnt and unburnt fuel stream elements.

The approach is elegant because atomic elements are conserved in chemical reactions. In turn, the mixture fraction is a conserved scalar quantity, and therefore its governing transport equation does not have a source term. Combustion is simplified to a mixing problem, and the difficulties associated with closing non-linear mean reaction rates are avoided.

Once mixed, the chemistry can be modeled as being in chemical equilibrium with the Equilibrium model, being near chemical equilibrium with the Steady Diffusion Flamelet model, or significantly departing from chemical equilibrium with the Unsteady Diffusion Flamelet model.

3.3.1.4 Discrete Phase Model (DPM):

The Lagrangian discrete phase model in ANSYS Fluent follows the Euler Lagrange approach. The fluid phase is treated as a continuum by solving the Navier-Stokes equations, while the dispersed phase is solved by tracking a large number of particles, bubbles, or droplets through the calculated flow field. The dispersed phase can exchange momentum, mass, and energy with the fluid phase.

The DPM performs Lagrangian trajectory calculations for dispersed phases (particles, droplets, or bubbles), including coupling with the continuous phase. Examples of multiphase flows include channel flows, sprays, sedimentation, separation, and cavitation.

3.3.2 User Defined Function (UDF):

A user-defined function, or UDF, is a C function that can be dynamically loaded with the ANSYS Fluent solver to enhance its standard features. For example:

- Customize boundary conditions, material property definitions, surface and volume reaction rates, source terms in ANSYS Fluent transport equations, source terms in user-defined scalar (UDS) transport equations, diffusivity functions, and so on.

- Adjust computed values on a once-per-iteration basis.
- Initialize of a solution.
- Perform asynchronous (on demand) execution of a UDF.
- Execute at the end of an iteration, upon exit from ANSYS Fluent, or upon loading of a compiled UDF library.
- Enhance post processing.
- Enhance existing ANSYS Fluent models (such as discrete phase model, multiphase mixture model, discrete ordinates radiation model).

UDFs are identified by a .c extension (for example, myudf.c). One source file can contain a single UDF or multiple UDFs, and multiple source files can be defined.

UDFs are defined using DEFINE macros provided by ANSYS Fluent. They are coded using additional macros and functions (also supplied by ANSYS Fluent) that access ANSYS Fluent solver data and perform other tasks.

Every UDF must contain the udf.h file inclusion directive (`#include "udf.h"`) at the beginning of the source code file, which enables both the definition of DEFINE macros and other ANSYS Fluent provided macros and functions, and their inclusion in the compilation process.

3.3.3 Solver:

3.3.3.1 Type:

There are two solvers types used in the Fluent program which are pressure based and density based type. The pressure-based solver is one of the powerful solvers employs an algorithm which belongs to the projection method. This model solves the nonlinear momentum and continuity equations to satisfy the conservation of mass and the conservation of momentum laws. To achieve the optimum solution for these equations iterations should be included.

- Pressure-based: Used for low speed and incompressible/slightly compressible flows. Pressure-Based enables the pressure-based Navier-Stokes solution algorithm.(the default)

- Density-based: Used for high speed and compressible flows. Density-Based enables the density-based Navier-Stokes coupled solution algorithm.

3.3.3.2 Velocity Formulation:

There are two types of velocity formulation methods in ANSYS Fluent –

- Absolute: used for stationary frame of reference.
- Relative: used for moving frame of references. This option is available only with the Pressure-Based solver.

3.3.3.3. Time:

Time contains options related to time dependence.

- Steady: used for time independent flow systems which do not change with time.
- Transient: used for time dependent flow systems which vary constantly with change in time.

3.3.3.4. 2D Space:

2D Space contains options available only when solving two-dimensional problems.

- Planar: indicates that the problem is two-dimensional.
- Axisymmetric: indicates that the domain is axisymmetric about the X axis. When Axisymmetric is enabled, the 2D axisymmetric form of the governing equations is solved instead of the 2D Cartesian form.
- Axisymmetric Swirl: specifies that the swirl component (circumferential component) of velocity is to be included in your axisymmetric model.

Chapter 4: Experimental Setup:

An experimentation was conducted to find the value of exhaust gas temperature at the end of a nozzle which is connected to the exhaust of the turbine. Specifications of the components used are as follows:

4.1 Components:

4.1.1 Turbocharger:

For the turbojet engine application the turbocharger is selected on the basis of compressor and turbine maps and based on calculations and requirements the turbocharger CZ STACONICE is selected which is sturdy and robust to withstand high speed, stresses and forces.

4.1.2 Combustion Chamber:

The combustion chamber was designed along with areas for primary, secondary and tertiary holes and their respective locations to achieve self-stabilizing flame, location of the igniter with the goal to achieve high combustion efficiency, reduction of visible smoke and reduction of oxides of nitrogen.

The shape of the combustion chamber is figured on the basis of the compressor outlet, the turbine inlet and the arrangement of the combustion chamber to the turbocharger.

Material used is SS310 seamless tubing of 1.5 mm thickness.

Optimized dimensions of the combustion chamber based on the feasibility and economic considerations:

- Overall length of the outer casing =300mm.
- Overall diameter of the outer casing =124mm.
- Length of the flame tube =253mm.
- Diameter of the flame tube =118mm.
- Optimum air gap between outer casing and flame tube =4mm.

4.1.3 Fuel Nozzle:

Spartech India Ltd. of capacity 4GPH with a hollow cone spray-to-spray fuel inside the combustion chamber.

4.1.4 Ignition System:

4.1.4.1 Transformer:

Neon transformer capacity 10,000 V which converts normal supply voltage to high voltage.

4.1.4.2 Spark Plug:

Bosch spark plug is used to create a high intensity spark.

4.1.5 Turbine:

The turbine is located at the rear of the turbocharger inside snail housing. The turbine used is of radial flow design. The snail housing is designed to increase the velocity of the inflowing air so that it strikes the turbine blades at high velocity. The gas temperature at this point is around 1800 ° F.

4.1.6 Propelling Nozzle dimensions:

- Nozzle tube diameter =82mm
- Throat diameter =58mm
- Cone angle = 8 degrees
- Cone length =85mm
- Straight length =325mm
- Overall length =410mm
- Tube thickness =2mm

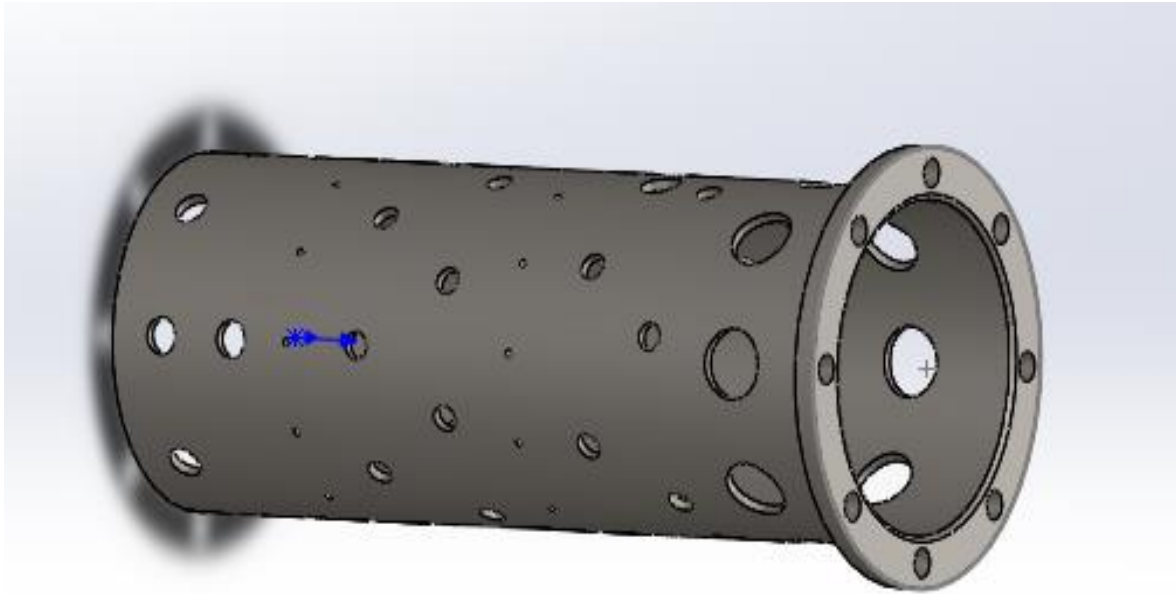


Figure 4.1: Solidworks Model of Flame Tube

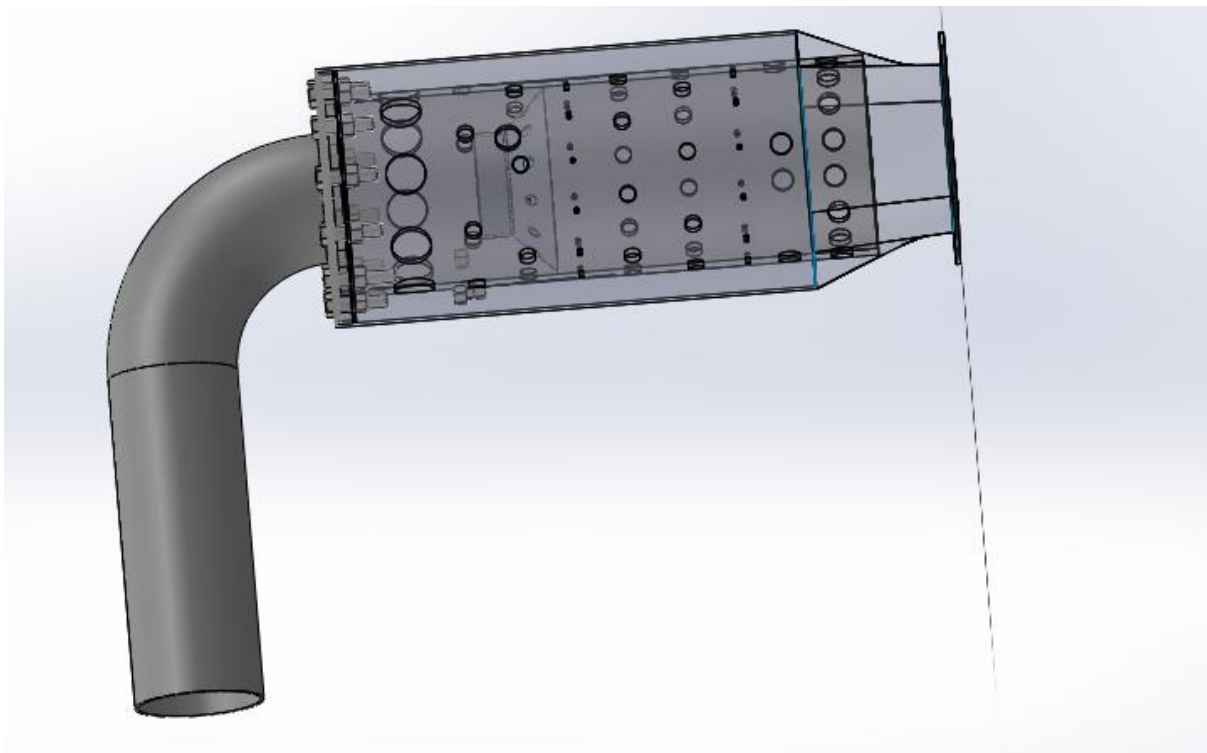


Figure 4.2: Solidworks Model of Complete Combustion Chamber Assembly

4.2 Measurement Instruments:

- **Pressure Gauge:**

Pressure solutions gauge with range 0-10Psi., uses oil pressure monitoring to measure pressure.

- **Control Panel:**

Consists of emergency stop switch, oil switch, ignition switch, fuel switch for switching ON/OFF of the engine system and controlling the process of combustion.



Figure 4.3: Experimental Setup of Turbojet Engine

4.3 Experimentation Results:

Pressure(Psi)	Pressure(bar)	RPM	Static Thrust (kgf)	EGT(K)
2	0.1378	40500	0.45	809.6
4	0.2756	43900	0.79	806.8
6	0.4134	45000	2.27	804
8	0.5512	46000	3.17	801.3
10	0.689	51500	4.53	801
12	0.8268	53000	6.12	801
14	0.9646	56500	7.12	804
16	1.1024	58120	7.26	809.7
18	1.2402	60000	8.16	815.2
20	1.3780	64000	8.98	820
22	1.5158	66500	9.53	823
24	1.6536	68200	11.33	826.3
26	1.7914	72000	13.1	829
28	1.9292	76100	14.5	829
30	2.067	78100	16.6	833

Table: Recorded Experimental Values of engine parameters

Chapter 5: Calculations

Calculation of Design Parameters

Data Assumptions:

$$\eta_c = 77\%$$

$$\eta_T = 78\%$$

$$\eta_{noz} = 85\%$$

$$\eta_{Tran} = 92\%$$

$$\eta_{comb} = 85\%$$

$$\gamma_a = 1.4$$

$$\gamma_g = 1.33$$

$$C_{pa} = 1.005 \text{ _KJ / Kg / K}$$

$$C_{pg} = 1.147 \text{ _KJ / Kg / K}$$

$$T_{amb} = T_0 = 293K$$

$$P_{amb} = P_0 = 0.98bar$$

$$\text{Compressor inlet temperature } T_{01} = 300 \text{ K}$$

$$\text{Turbine outlet temperature } T_{04} = 880 \text{ K}$$

$$\Delta P_{comb} = 0.2bar$$

$$Q_f = 43 \text{ _MJ}$$

$$\text{Experimental data: } P_{02} = 2.0685 \text{ bar}$$

Other Variables:

1. Compressor outlet temperature $T_{02} = ?$

2. Turbine inlet temperature $T_{03} = ?$

3. Turbine pressure ratio $\frac{P_{03}}{P_{04}} = ?$

4. Nozzle outlet temperature $T_{05} = ?$

5. Jet speed $C_5 = ?$

6. Intake air mass flow rate $\dot{m}_a = ?$
 $\frac{\dot{m}_a}{\dot{m}_f} = ?$
7. Air-fuel ratio \dot{m}_f
8. Fuel flow rate $\dot{m}_f = ?$
9. Exhaust gases mass flow rate $\dot{m}_g = ?$

For T-S diagram refer Figure 2.2

1. Diffuser

Assume diffuser is absent in the turbocharger unit, hence there is no ramming therefore air directly enters the compressor eye at atmospheric pressure at temperature 300K

2. Compressor

Air enters the compressor at temperature 300K and pressure 0.98 bar. Pressure at the compressor exit is taken from the experimental readings to be 2.0685 bar

$$\frac{T_{02'}}{T_{01}} = \left(\frac{P_{02'}}{P_{01}} \right)^{\frac{\gamma_a - 1}{\gamma_a}}$$

$$T_{02'} = T_{01} \times \left(\frac{P_{02'}}{P_{01}} \right)^{\frac{\gamma_a - 1}{\gamma_a}}$$

$$= 300 \times 2.0685^{0.2857}$$

$$= 371.37 \text{ K}$$

$$\therefore T_{02'} = 371.37 \text{ K}$$

Compressor efficiency :

$$\eta_c = \frac{T_{02'} - T_{01}}{T_{02} - T_{01}}$$

$$T_{02} - T_{01} = \frac{T_{02'} - T_{01}}{\eta_c}$$

$$= \frac{371.37 - 300}{0.77}$$

$$= 92.68 \text{ K}$$

$$\therefore T_{02} = 392.68 \text{ K}$$

3. Turbine:

All the work done by the turbine is used to drive the compressor therefore

Work done by turbine = Work done on compressor

$$\eta_{T_{ran}} \times C_{pg} (T_{03} - T_{04}) = C_{pa} \times (T_{02} - T_{01})$$

$$T_{03} - T_{04} = \frac{C_{pa} \times (T_{02} - T_{01})}{\eta_{T_{ran}} \times C_{pg}}$$

$$= \frac{1.005 \times 92.68}{0.92 \times 1.147}$$

$$= 88.26 \text{ K}$$

Now,

$$T_{04} = 880 \text{ K}$$

$$\therefore T_{03} = 968.26 \text{ K}$$

Turbine efficiency:

$$T_{03} - T_{04} = \eta_T \times T_{03} \times \left(1 - \frac{1}{r_t^{\frac{\gamma_g - 1}{\gamma_g}}} \right)$$

$$88.26 = 0.78 \times 968.26 \times \left(1 - \frac{1}{r_t^{\frac{1.33 - 1}{1.33}}} \right)$$

$$\therefore r_t = 1.6501$$

$$r_t = \frac{P_{03}}{P_{04}} = \frac{P_{02} - \Delta P_{comb}}{P_{04}}$$

$$\begin{aligned} \therefore P_{04} &= \frac{P_{02} - \Delta P_{comb}}{r_t} \\ &= \frac{2.0685 - 0.2}{1.6501} \\ &= 1.1323 \end{aligned}$$

$$\therefore P_{04} = 1.1323 \text{ bar}$$

4. Nozzle:

Exhaust gases from turbine outlet enters the propelling nozzle at pressure P_{04} and temperature T_{04} .

$$\frac{T_{04}}{T_5} = \left(\frac{P_{04}}{P_5} \right)^{\frac{\gamma_g}{\gamma_g - 1}}$$

$$= \left(\frac{1.1323}{1} \right)^{\frac{1.33 - 1}{1.33}}$$

$$\therefore T'_5 = \frac{880}{1.0313} = 853.28 \text{ K}$$

Nozzle efficiency:

$$\eta_{noz} = \frac{T_{04} - T_{5'}}{T_{04} - T_5}$$

$$T_{04} - T_5 = \frac{T_{04} - T_{5'}}{\eta_{noz}}$$

$$= \frac{880 - 853.28}{0.85}$$

$$= 26.72 \text{ K}$$

$$\therefore T_5 = 848.56 \text{ K}$$

Jet speed:

$$\frac{C_5}{2C_{pg}} = T_{04} - T_5$$

$$= 38.25$$

$$C_5 = \sqrt{2 \times C_{pg} \times \Delta T}$$

$$= \sqrt{2 \times 1147 \times 26.72}$$

$$= 268.55 \text{ m/s}$$

$$\therefore C_5 = 268.55 \text{ m/s}$$

Thrust:

$$F = \dot{m}_a \times (C_j - C_i)$$

$$C_i = 0$$

$$\therefore F = \dot{m}_a \times (C_j)$$

$$m_a = \frac{F}{C_j} = \frac{166}{268.55} = 0.6181 \text{ kg/s}$$

$$\therefore m_a = 0.6181 \text{ kg/s}$$

Air-Fuel ratio:

$$\begin{aligned} h_{03} - h_{02} &= \eta_{comb} \times \dot{m}_f \times C.V \\ &= \frac{C_{pa} \times (T_{03} - T_{02})}{\eta_{comb} \times C.V} \\ \therefore \frac{m_f}{m_a + m_f} &= \frac{1.005 \times (968.26 - 392.68)}{0.85 \times 43000} \\ \therefore m_f &= 0.01136 \text{ kg/s} \end{aligned}$$

Therefore, Air-Fuel ratio is given by = 54.36:1

By pass ratio:

Area of inlet to combustion chamber = 454 mm²

Total area of holes for bypassing the air = 2280 mm²

Thus, the mass flow rate to the inlet and bypass is divided in the corresponding ratio of their areas.

$$\therefore \text{Bypass ratio} = \frac{2280}{454} = 5.022$$

$$\therefore m_{\text{bypass}} = \frac{2280}{454 + 2280} \times 0.6181 = 0.5155 \text{ kg/s}$$

The bypass air is further divided into primary, secondary and tertiary holes.

There too the division is based on areas.

A_{zone} = no of holes x area of each hole

$$A_{\text{primary}} = 12 \times \frac{\pi}{4} \times 7.5^2 = 530.14 \text{ mm}^2$$

$$A_{\text{secondary}} = 16 \times \frac{\pi}{4} \times 7.5^2 + 24 \times \frac{\pi}{4} \times 2^2 = 782.25 \text{ mm}^2$$

$$A_{\text{tertiary}} = 12 \times \frac{\pi}{4} \times 10^2 = 942.47 \text{ mm}^2$$

Therefore, the mass flow is divided in the following ratio:

Primary : Secondary : Tertiary = 0.235 : 0.347 : 0.418

$$\therefore m_{primary} = 0.235 \times 0.5155 = 0.1211 \text{ kg/s}$$

$$\therefore m_{secondary} = 0.347 \times 0.5155 = 0.1789 \text{ kg/s}$$

$$\therefore m_{tertiary} = 0.418 \times 0.5155 = 0.2155 \text{ kg/s}$$

Chapter 6: Numerical Procedure

Aspects included during numerical procedure

- Complicated fuel injection system
- Primary zone fuel air ratio, secondary zone fuel air ratio
- Length and diameter of the combustor
- Operating pressure
- Holes distribution throughout the chamber, diameters of these holes
- Total pressure at the annular space from where secondary and dilution air flow to the chamber.

6.1 ANSYS Solution Setup: ANSYS FLUENT

6.1.1 General

1. Mesh

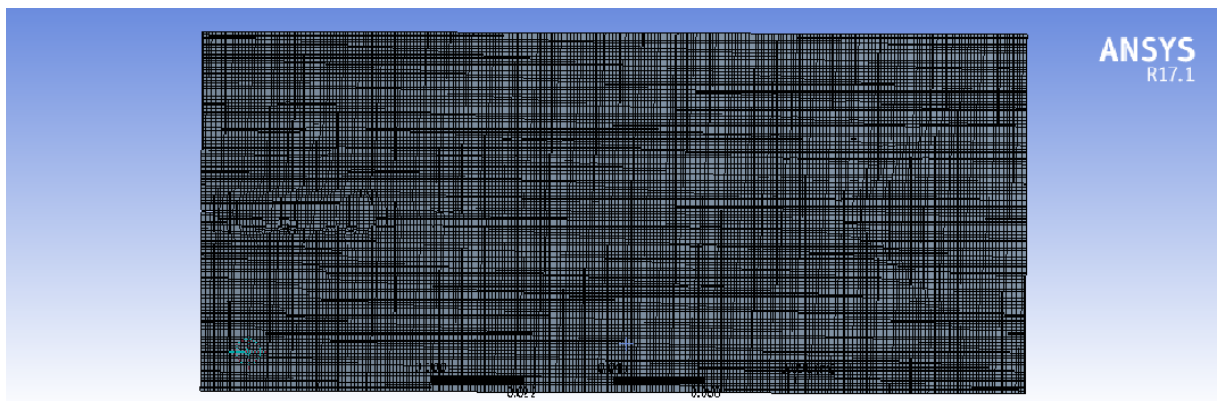


Figure 6.1: 2D Mesh Model

Major mesh setting variables include the following-

1. Defaults
 - Physical reference CFD
 - Solver Preference FLUENT
 - Relevance 0

2. Sizing

- Use Advanced Size Function On: Curvature
- Relevance Center Coarse
- Smoothing Medium
- Span Angle Center Fine
- Minimum Size Default (1.34794-004m)
- Maximum Size Default (1.34794-002m)
- Minimum Edge Length 5.e-004m

3. Face Sizing

- Suppressed No
- Type Element Size
- Element Size 8.e-004m.

Mesh Quality:

- Minimum Orthogonal Quality=0.732763
(Orthogonal Quality ranges from 0 to 1, where 0 corresponds to low quality)
- Maximum Ortho Skew=0.196773
(Ortho Skew ranges from 0 to 1, where 1 corresponds to low quality)
- Maximum Aspect Ratio=3.64805

2. Solver

- **Type**

(a) Pressure-based

(b) Density-based

Selecting pressure-based Navier-Stokes coupled algorithm assuming air to be incompressible fluid and velocity of flow during the experiment was in the range of 40-70m/s. Mach number remains less than 0.3 as the speed of sound at a temperature of 410 K is 405 m/s. Thus, flow was assumed to be incompressible and pressure based solver was used.

- Velocity formulation

(a) Absolute

(b) Relative

Selecting absolute as the frames are not moving reference frames.

- Time

(a) Steady

(b) Transient

Selecting transient as time dependant analysis.

- 2D Space

(a) Planar

(b) Axisymmetric

(c) Axisymmetric Swirl

Selecting Planar as Model is in 2D plane.

Gravitational Acceleration Y (m/s^2) = -9.81

6.1.2 Models

- Multiphase model: Off
- Energy equation: On
- Viscous model: Realizable k-epsilon (2 eqn) with enhanced wall treatment having pressure and gradient effects.

In this simulation, the Realizable κ - ϵ model has been used because of his several advantages over other models: economy, strength, and accuracy.

- Radiation model: P1
- Species model: Non premixed combustion

PDF Table Creation

- Chemistry
 - (a) State Relation: Chemical Equilibrium
 - (b) Energy Treatment :Non-Adiabatic
- Boundary
 - Species
 - Fuel : $\text{C}_{12}\text{H}_{23}$ =1
 - Oxidizer : O_2 =0.21008

$$N_2=0.78992$$

Temperature

- Fuel :300K
- Oxidizer : 410K

Specify Species in Mole Fraction

- Discrete Phase Model

Interaction with Continuous Phase

Injection:

Injection Type: Group

Number of Streams: 5

Particle Type: Droplet

Set Injection Properties

Injection Name: injection-0 Injection Type: group Number of Streams: 5

Particle Type: ☐ Massless ☐ Inert ☒ Droplet ☐ Combusting ☐ Multicomponent Laws: ☐ Custom

Material: kerosene-liquid Diameter Distribution: linear Oxidizing Species: Discrete Phase Domain: none

Evaporating Species: c12h23 Devolatilizing Species: Product Species:

Point Properties Physical Models Turbulent Dispersion Parcel Wet Combustion Components UDF Multiple Reactions

Variable	First Point	Last Point
X-Position (m)	0.003	0.003
Y-Position (m)	0.04475	0.04475
X-Velocity (m/s)	0	47.45
Y-Velocity (m/s)	0.0001	0.0001
Diameter (m)	1e-06	1e-06
Temperature (k)	300	300
Flow Rate (kg/s)	0.01136	0.01136

Figure 6.2: Injection Properties

6.1.3 Material: kerosene-air mixture

6.1.4 Boundary conditions:

1) Air inlet:

Mass Flow Inlet

- Mass flow rate: 0.1026 kg/s
- Initial gauge pressure: 108215 pascal
- Direction: Normal to boundary
- Turbulent intensity: 15%
- Hydraulic Diameter: 0.0495m.
- Total temperature: 410K

2) Fuel inlet:

Mass Flow Inlet

- Mass flow rate: 0.01136 kg/s
- Initial gauge pressure: 551581 pascal
- Direction: Normal to boundary
- Turbulent intensity: 5%
- Turbulent Viscosity Ratio:10
- Total temperature: 300

3) Primary air inlet:

Mass Flow Inlet

- Mass flow rate: 0.1211 kg/s
- Initial gauge pressure: 108215 pascal
- Direction: Normal to boundary
- Turbulent intensity: 15%
- Hydraulic Diameter: 0.013m.
- Total temperature: 410K

4) Secondary air inlet:

Mass Flow Inlet

- Mass flow rate: 0.1789 kg/s
- Initial gauge pressure: 108215 pascal

- Direction: Normal to boundary
- Turbulent intensity: 15%
- Hydraulic Diameter: 0.015 m.
- Total temperature: 410K

5) Tertiary air inlet:

Mass Flow Inlet

- Mass flow rate: 0.2155 kg/s
- Initial gauge pressure: 108215 pascal
- Direction: Normal to boundary
- Turbulent intensity: 15%
- Hydraulic Diameter: 0.01732 m.
- Total temperature: 410K

6) Spark Plug:

Mass Flow Inlet

- Mass flow rate: UDF spark plug
- Thermal: Total Temperature=1173K

UDF Code for Spark Plug

```
#include "udf.h"

DEFINE_PROFILE(sparkplug,th,i)

{

face_t f;

begin_f_loop(f,th)

{

real flow_time = RP_Get_Real("flow-time");
```

```

if(flow_time<=0.002)

{

F_PROFILE(f,th,i)=0.637;


}

else

{

F_PROFILE(f,th,i)=0;

}

}

end_f_loop(f,th)

}

```

7) **Outlet**

Pressure Outlet

- Gauge Pressure: 81603 pascal
- Backflow Turbulent Intensity: 5%
- Backflow Hydraulic Diameter: 0.09m.
- Backflow Total Temperature: 300K.

Chapter 7: Results

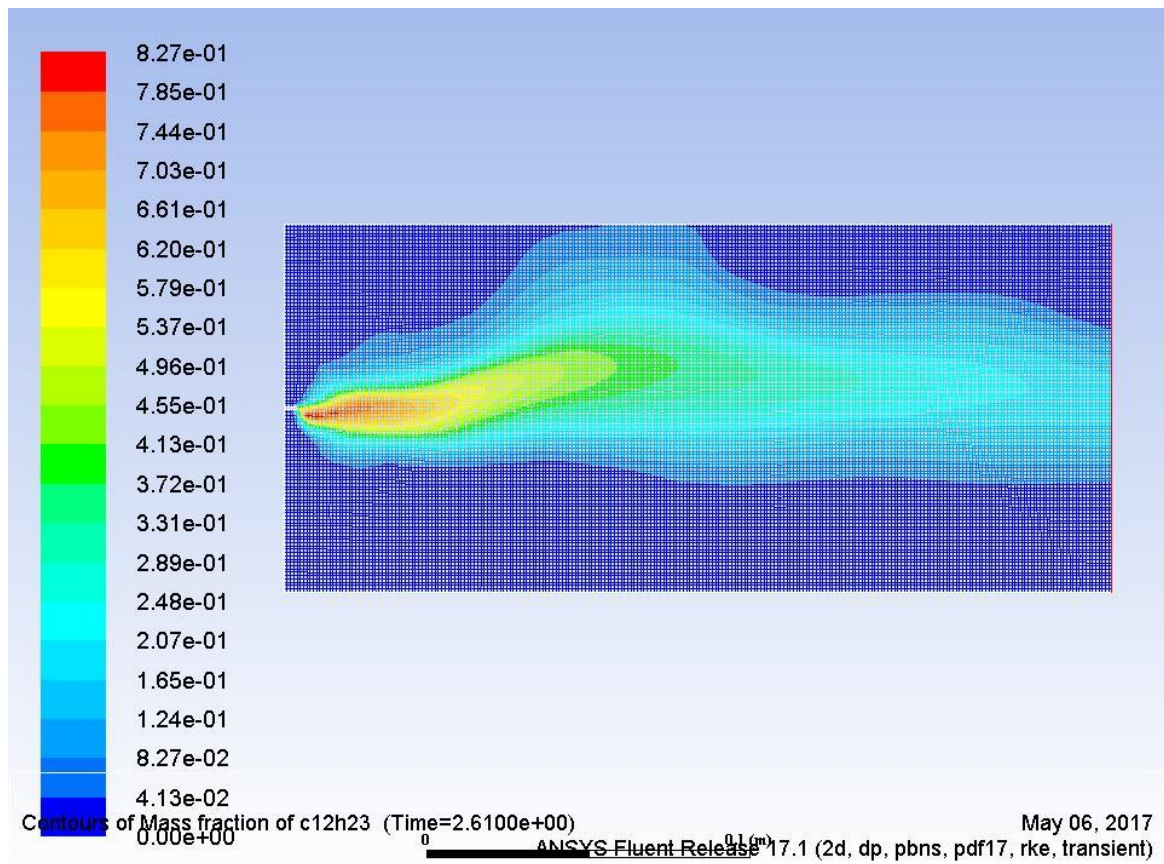


Figure 7.1: Contour of mass fraction of Kerosene

Maximum mass fraction of kerosene = 0.8266

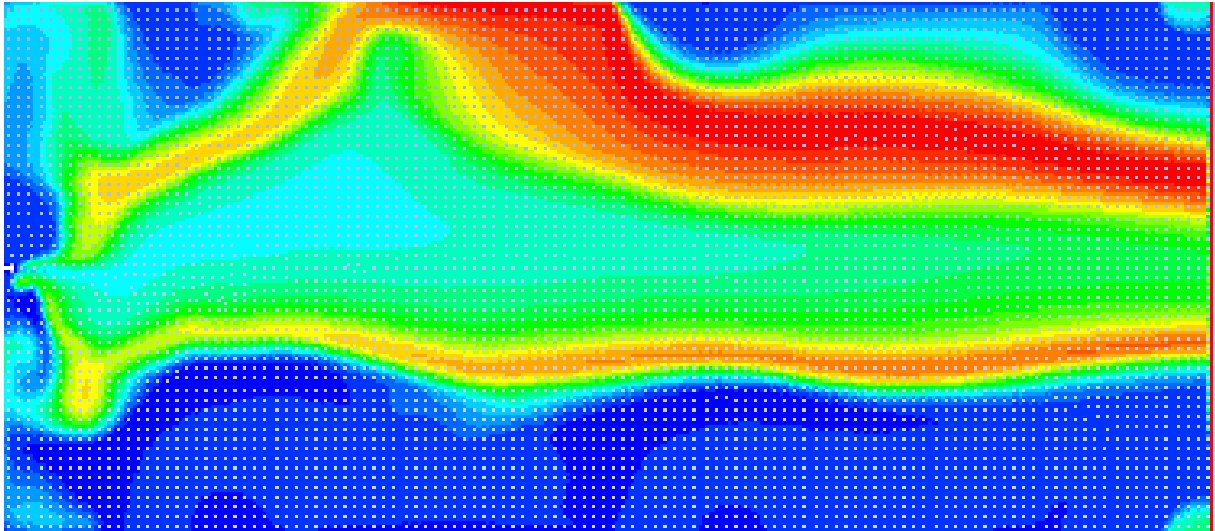


Figure 7.2: Temperature profile at Spark ignition

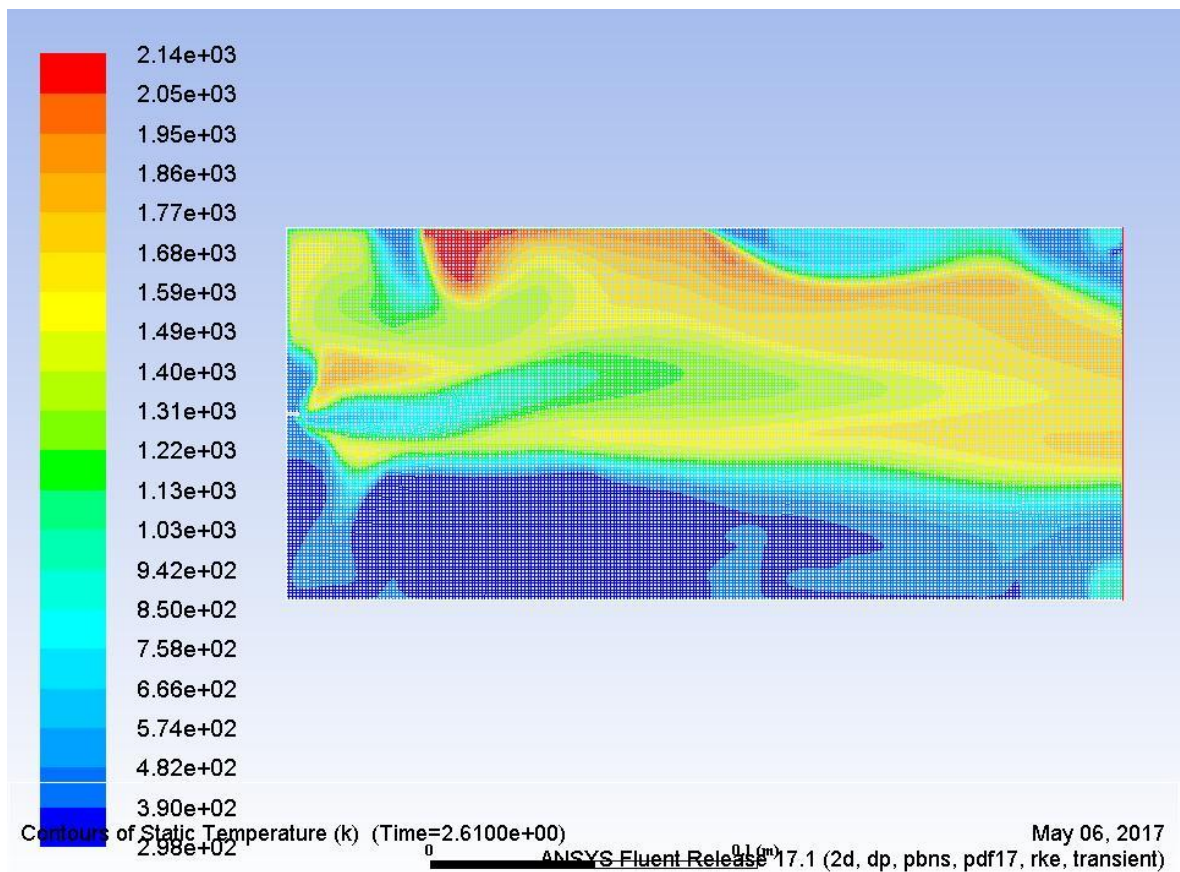


Figure 7.3: Contour of Static Temperature (K)

Static Temperature:

Minimum value = 298 K

Maximum value = 2138.927 K

Mass weighted average static temperature at outlet = 1070 K

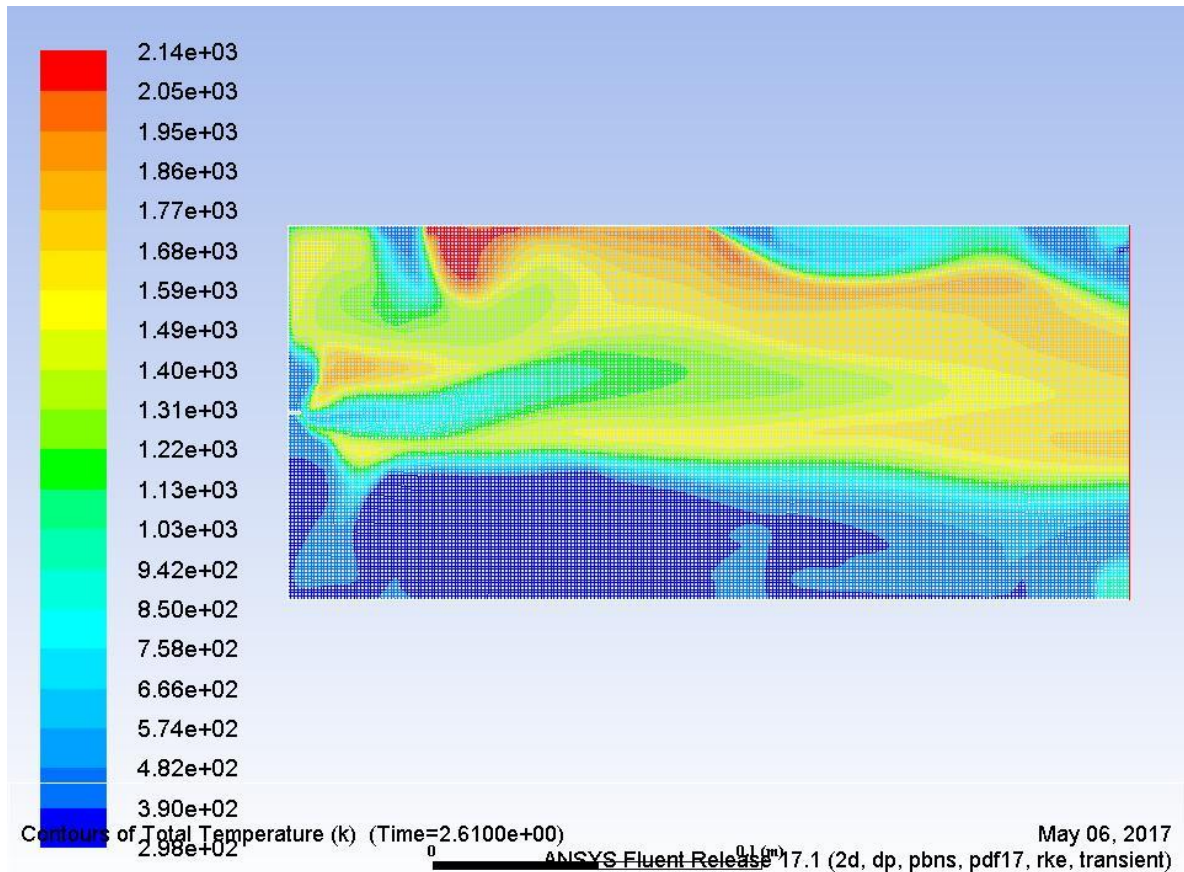


Figure 7.4: Contour of Total Temperature (K)

Total Temperature:

Minimum value = 298 K

Maximum value = 2138.927 K

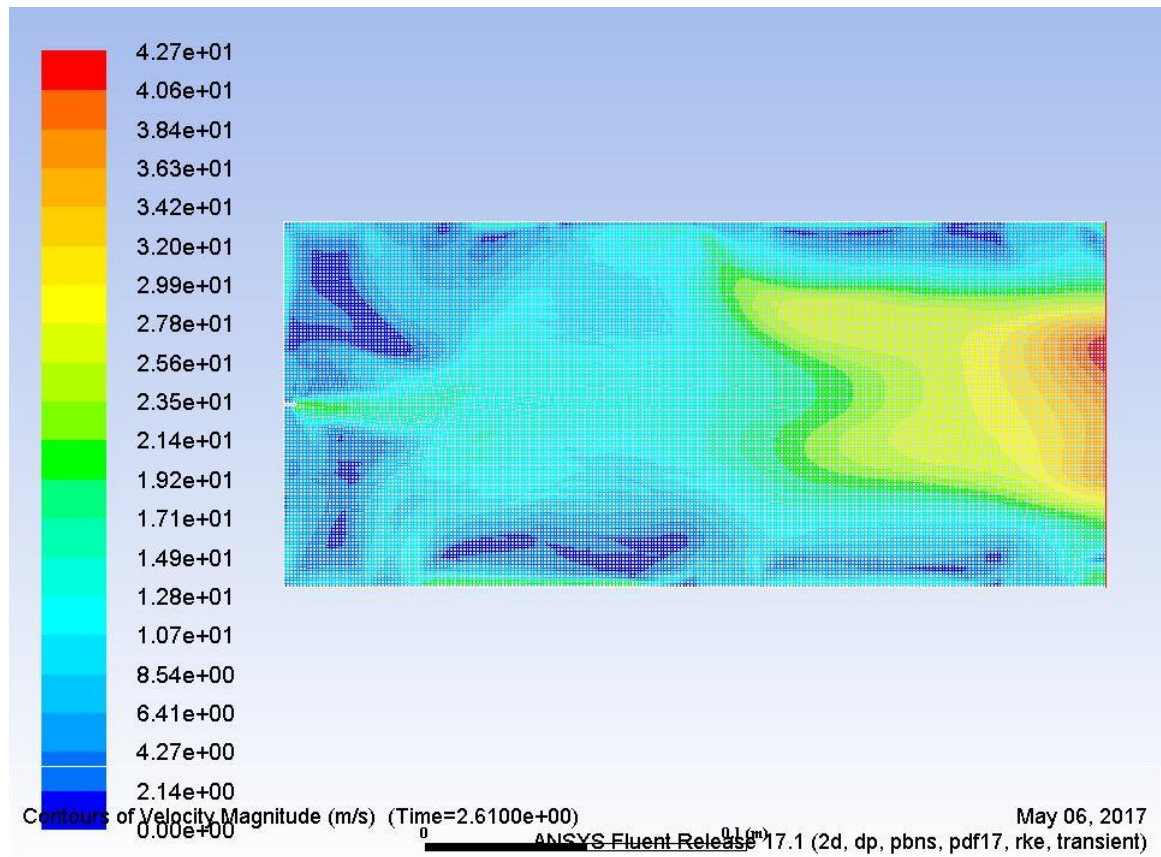


Figure 7.5: Contour of Velocity Magnitude (m/s)

Velocity Magnitude:

Minimum value = 0 m/s.

Maximum value = 42.71 m/s.

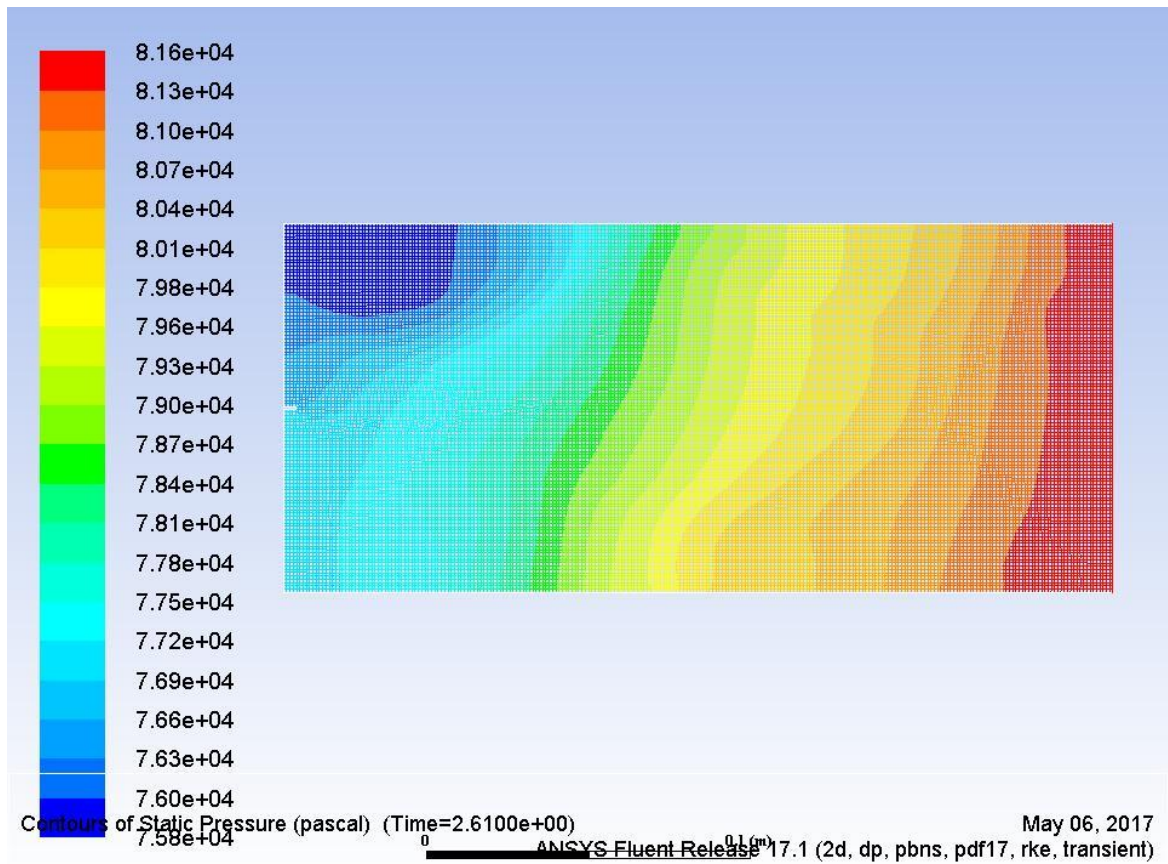


Figure 7.6: Contour of Static Pressure (pascal)

Static Pressure:

Minimum value = 75752 Pa

Maximum value = 81605 Pa

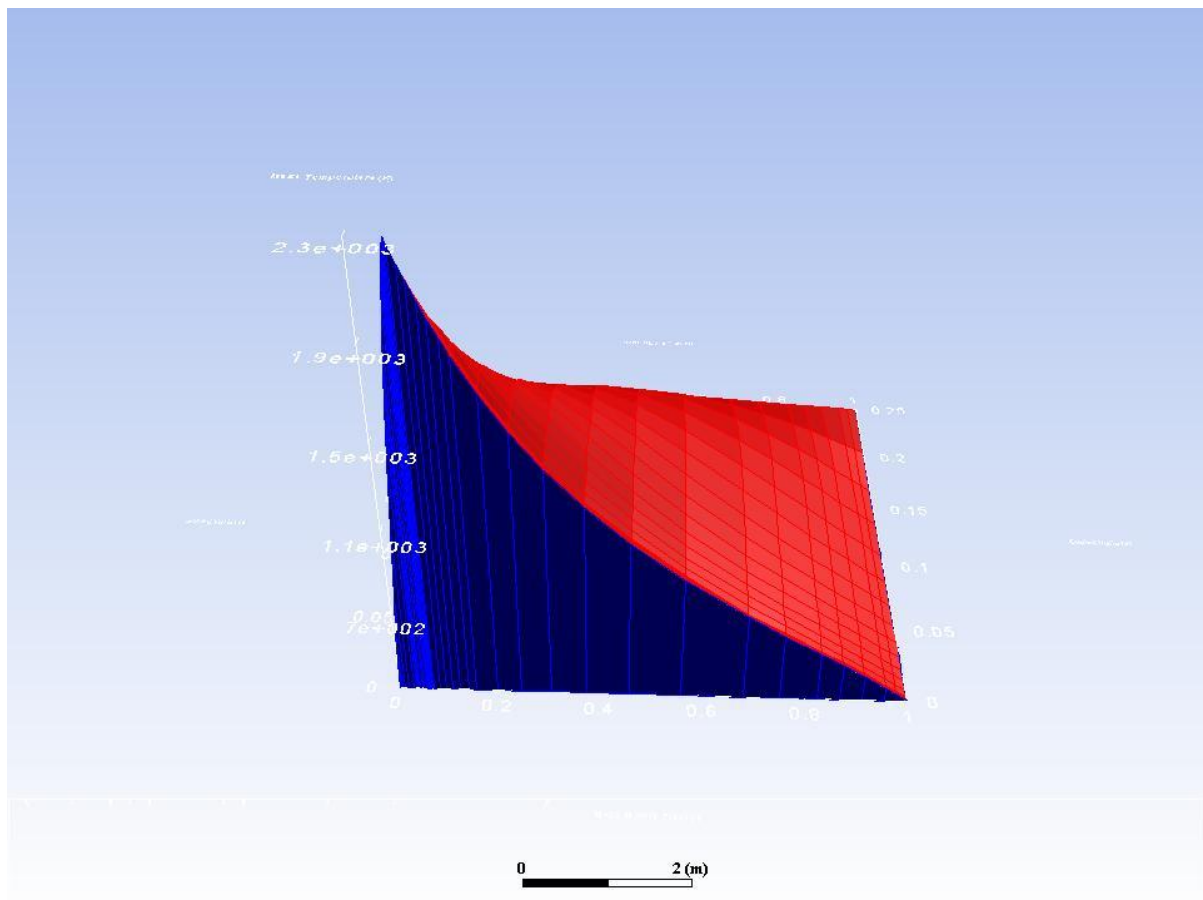


Figure 7.7: PDF Table

Adiabatic flame temperature (From PDF table) = 2323 K

Adiabatic flame temperature (online calculator) [7] = 2394.15 K

Chapter 8: Conclusion

It has been observed that CPU time increases proportionally with number of cells in the computational domain. Increases in number of species affects the computational time nonlinearly.

The temperatures and pressure profiles of various models are tested in a virtual medium to avoid generation of extremely high temperatures, which can cause damage to the turbine blades.

Adiabatic flame temperature calculated using calculator based on theoretical equation [7] was found to be 2394.15 K, while Numerical value obtained is 2323 K. Thus, the error is approximately 3% i.e. within acceptable range.

Different contour plot of different variables are visualized to understand the physics and chemistry of the model.

Value of temperature for the end gas region are found experimentally and verified with the numerical solution using ANSYS Fluent which are within acceptable range and error for EGT is found with respect to theoretical value and experimental value. Thus, model can be used with few improvements for analysing few other similar models too.

% Error with respect to theoretical value =

$$\left| \frac{\text{theoretical value of EGT} - \text{numerical value of EGT}}{\text{theoretical value of EGT}} \right| = \left| \frac{880 - 1042}{880} \right| = 18.41 \%$$

% Error with respect to experimental value =

$$\left| \frac{\text{experimental value of EGT} - \text{numerical value of EGT}}{\text{experimental value of EGT}} \right| = \left| \frac{848.56 - 1042}{848.56} \right|$$

= 22.79 %

Chapter 9: Future Scope

- This model can be modified so as to predict the emissions of various pollutants and gases which are harmful to the atmosphere.
- The location of the spark plug and its timing can be varied for efficient burning of the fuel in the combustion chamber.
- The quantity of fuel and its inlet location can also be varied and checked until the most feasible solution is obtained.
- The amount of air flowing as bypass could be adjusted, and the locations and the number of primary, secondary, tertiary holes can be changed for effective cooling of the combustion chamber and the turbine blades.

Chapter 10: References

- [1] Gas Turbines; V Ganesan; Tata Mcgraw-Hill Publishing Company Limited; New Delhi; 2003; 1st Edition.
- [2] ANSYS Fluent Theory Guide, Release 15.0, November 2013.
- [3] ANSYS-FLUENT-14.0-Combustion Guide
- [4] CFD Modelling Combustor, Master's Thesis in Solid and Amir Khodabandeh, Department of Applied Mechanics Division of Fluid Dynamics CHALMERS UNIVERSITY OF TECHNOLOGY Göteborg, Sweden 2011 Master's thesis 2011:47
- [5] Recent progress and challenges in fundamental combustion research, Yiguang Ju†, Department of Mechanical and Aerospace Engineering, Princeton University, New Jersey, USA.
- [6] Modeling Turbulent Combustion CEFRC Combustion Summer School Prof. Dr. Ing. Heinz Pitsch, RWTHAACHEN University.
- [7] <http://elearning.cerfacs.fr/>
- [8] P. Cheng. "Two-Dimensional Radiating Gas Flow by a Moment Method". AIAA Journal. 2. 1662–1664. 1964.
- [9] Fluent Incorporated.
- [10] Dagaut, P., Cathonnet, M.: The ignition, oxidation, and combustion of kerosene: A review of experimental and kinetic modeling. Progress in Energy and Combustion Science 32, 48–92 (2006)
- [11] Pitz, W., Mueller, C.: Recent progress in the development of diesel surrogate fuels. Progress in Energy and Combustion Science 37, 330-350 (2011)
- [12] Violi, A., Yan, S., Eddings, E., Granata, S.: Experimental Formulation and Kinetic model for JP-8 surrogate mixtures. Combustion Science and Technology 174(11&12), 399-417 (2002)
- [13] Humer, S., Frassoldati, A., Granata, S., Faravelli, T., Ranzi, E.: Experimental and kinetic modeling study of combustion of JP-8, its surrogates and reference components in laminar nonpremixed flows. Proceedings of the Combustion Institute 31, 393–400 (2007)
- [14] <http://web.mit.edu/16.unified/www/FALL/thermodynamics/notes/node111.html>

Chapter 11: Acknowledgment:

The learning curve is a continuous one. Each day, we imbibe something new, take in new concepts, and are exposed to new things. The process of learning would be an impossible one, if not for a strong support system as well from our Guide and mentor Dr. Arvind Deshpande.

As a project guide, he has been responsible for constantly pushing us towards excellence. We would like to acknowledge particularly, his supporting our actions, solving our doubts at every stage, providing us with invaluable advice, and most importantly, the knowledge bestowed upon us by him. His friendly nature has been a real boon for us, for he has been extremely approachable and helpful- the ideal qualities one would look for in an inspiring guide. Of course, his passion for the field and enthusiasm shown towards our findings made our work environment a very positive one.

We would also like to thank Mr. Flevian Gonsalves and Mr. Pramod Pawar for being such helpful seniors and for providing a basis for experimentation and validation of our project. Their constant support and guidance has been of immense help to us.

We also extend our thanks to the teaching and non-teaching staff at Mechanical Engineering Department of Veermata Jijabai Technological Institute, Mumbai led by the departmental head. Dr. N. P. Gulhane for being ever so patient with us and providing us access to educational and infrastructural facilities without which this dissertation would have been incomplete.

Finally, we would also like to thank Prof. S. H. Kulkarni, our professor at V.J.T.I., for cultivating the deep rooted concepts of Heat Transfer and Internal Combustion Engines within us. She has always supported us as students, urged us to think innovatively, and her watchful eye has led many a student from going astray. We thank her for her constant motivation and guidance, which has been very special to us over the past two years.

Satyam Ashokrao Avhad
131020004

Sakina Moiz Tinwala
131021039

Date:

Place: