# DESIGN AND DEVELOPMENT OF TORCH IGNITOR PROJECT REPORT

**[1]AKSHAY V V, [2]Dr. Vasanth Kumar, [3]Mr. Dinesh Kumar**

[2]HOD, [3]Assistant professor
Department of Aerospace Engineering
SCHOOL OF AERONAUTICAL SCIENCES
HINDUSTAN INSTITUTE OF TECHNOLOGY AND SCIENCE
PADUR, CHENNAI-603103

***Abstract-*** **The purpose of this project is to design and develop a torch ignition system in order to implement it into a Gas Turbine engine. This system is of fundamental importance to a reliable operation of a gas turbine engine combustor. Various designs of torch ignitor are taken into consideration and most suitable design was selected with non-reacting flow analysis to find the development of the flow, pressure inside the chamber, velocity inside the chamber, throat and exit section of the ignitor. The design of torch ignitor simulation using K-epsilon, realizable, standard wall function with Ansys fluent software was done. Ignitor and fuel nozzles are selected on the basis of design requirements. Reacting flow analysis was done using non premixed steady diffusion flamelet with non-adiabatic energy treatment. The model simulates discrete phase model with injection of fuel done in the form of droplet with pressure swirl injection.**

**CHAPTER 1**

**INTRODUCTION**

**GAS TURBINE ENGINE**

A gas turbine engine, also known as a jet engine, is a type of internal combustion engine that mainly powers aircraft but also has uses in power generation, marine propulsion, and industrial processes.

The Brayton Cycle governs the operation of a gas turbine motor.

Gas turbine engines operate by drawing air through an intake, compressing it in a compressor, and mixing the compressed air with fuel in the combustion chamber. The fuel-air mixture is ignited with an ignitor, and the resulting hot gases expand quickly, driving a turbine that turns a shaft, which powers the compressor and, occasionally, an external load such as a propeller or generator. After that, the exhaust gases are expelled through a nozzle, creating thrust that propels the aircraft forward.

Gas turbine engines are classified into two types: turbojet engines and turbofan engines. The basic type of gas turbine engine is a turbojet engine, which consists of a compressor, combustion chamber, turbine, and nozzle. Turbofan engines, on the other hand, have a fan in front of the compressor that provides extra thrust by bypassing some of the air around the combustion chamber, resulting in higher fuel efficiency and lower noise

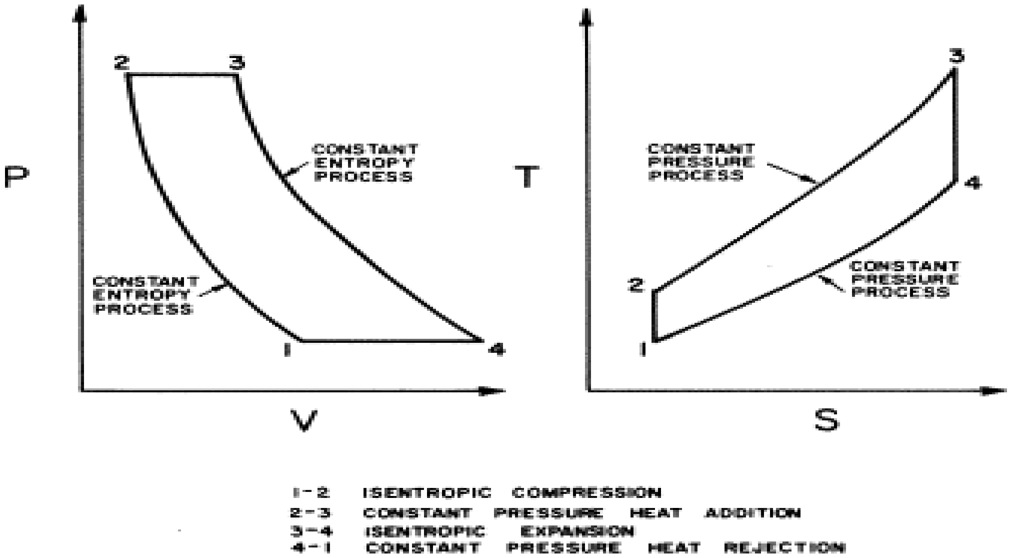



*Figure 1: Brayton cycle*

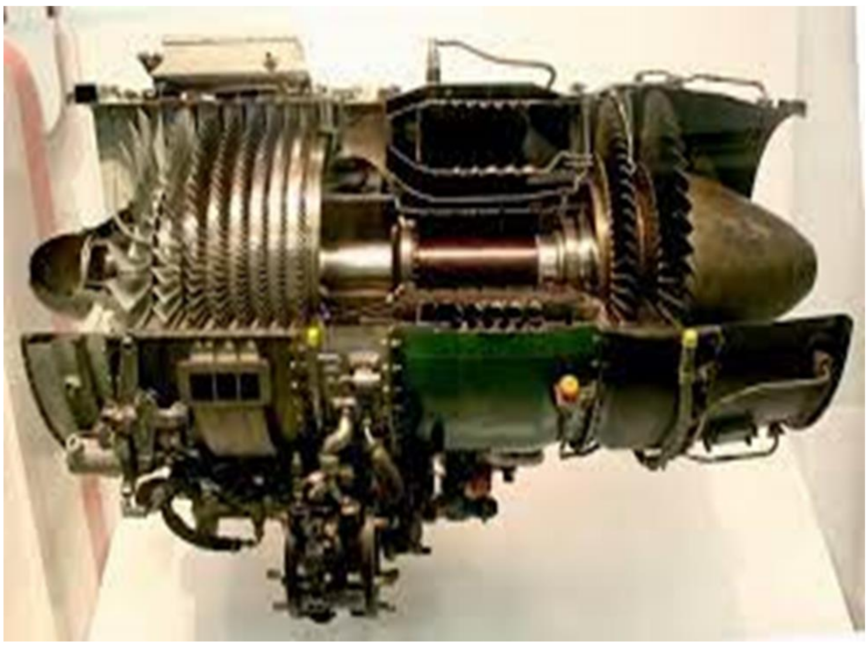


*Figure 2: Gas turbine engine*

A gas turbine engine's combustion chamber is a crucial component that is responsible for the combustion process that generates hot gases that drive the engine's turbines. The combustion chamber is situated between the engine's compressor and turbine sections.

Combustors are engineered to burn fuel and air in a controlled way to generate the desired amount of thrust.

In an aviation combustor, the combustion process begins with the intake of a mixture of fuel and air, which is then ignited by a spark or any other ignition source. The resulting combustion reaction produces high-temperature gases that quickly expand, resulting in a high-pressure, high-velocity jet of exhaust gases that aids in propelling the aircraft forward.

In aircraft engines, there are several kinds of combustion chambers, including can-annular, annular, and can combustors.

The most prevalent types of combustors used in modern aircraft engines are can-annular and annular.

Aircraft combustion chamber design is essential for optimizing engine performance and lowering emission

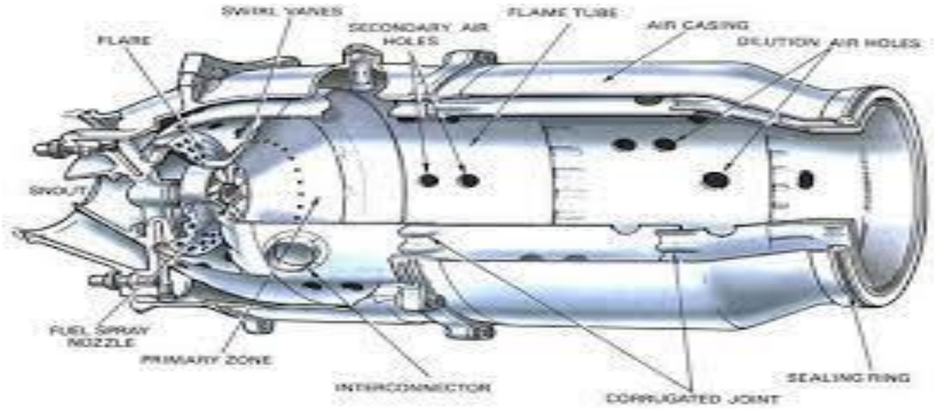



*Figure 3: Combustion chamber*

**COMBUSTION**

Combustion is a chemical reaction in which fuel interacts quickly with an oxidizer, releasing heat.

Although fuel can be solid, liquid, or gas, it is typically a liquid in airplane propulsion. For airplanes, the oxidizer can be a solid, liquid, or gas, but it is typically a gas (air).

Even though the engine includes fuel and air, combustion requires an extra source of heat. Because heat is needed to start combustion and is also a product of combustion, combustion occurs very quickly. Furthermore, once combustion begins, we do not need to provide a heat source because the heat of combustion will aid in subsequent burning

With a speed of around 150 m/s, compressed air reaches the combustion chamber.

Its initial purpose is to slow down the compressed air stream before increasing its atmospheric pressure. Because of jet fuel's high flame speed, even a decreased flow velocity of 24.4 m/s would extinguish any flame. As a result, a combustion region with much lower axial velocity air is needed to sustain combustion over a broad range of operating conditions.

The air-to-fuel (AF) ratio varies between 45:1 and 130:1 in typical operation, but kerosene-based fuels are most efficiently burned at an AF ratio of around 15:1.

Only a small portion of the air entering the combustion chamber participates in the combustion process, which is known as the main combustion zone. Around 20% of the air is directed straight into the chamber's core. There are swirl vanes at the site of entry that cause the incoming air to whirl, promoting improved air circulation and mixing throughout the chamber. The remaining 80% of the air mass is cooled by being passed between the chamber liner and its casing.

The liner is perforated, allowing up to 20% of the main flow to reach the primary combustion zone. The air that passed through the swirl vanes and these openings creates a small recirculation zone with low flow rates.

By the end of the primary zone, the temperature of the combustion products can approach 1800-2000°C and must be cooled before entering the turbine. To accomplish this, the previously unused air (approximately 60%) is progressively introduced into the chamber. One could say that roughly half of the secondary flow is used to chill the combustion gases and the other half is used to cool the combustion lining and casing. It is also important to note that combustion should be finished before introducing the coolest stream of air at the end of the combustor; otherwise, the combustion efficiency will drop dramatically due to the low temperature.

At starting, spark plugs are typically used to initiate combustion. When air flows through the core and the temperature increases, the combustion becomes self-sustaining, and the plugs are no longer needed. Combustion chamber design varies significantly from engine to engine in a variety of ways, including how fuel is injected, how cooling is realized, and how flow is managed.

Even though the engine contains fuel and air, combustion does not occur because there is no source of heat. Since heat is both required to start combustion and is itself a product of combustion, combustion takes place very rapidly. Also, once combustion gets started, we don't have to provide the heat source because the heat of combustion will keep things going.

But initially we have to provide and external heat source or energy source to initiate combustion process, for which we make use of **Ignitors**. The fuel in the combustor is ignited using it, and it is then turned off.

The two main types of jet engine igniting systems are:

**Capacitor type** - which uses high-energy, extremely high-temperature sparks generated by a condenser discharge

**Induction type** - which produces high-tension sparks using standard induction coils.

Also **glow plug** is used which has the benefit of not producing the same kind of electromagnetic radiation as the capacitor ignition system, which eliminates the need for a filter to prevent interference with the aircraft's electronic components

**IGNITION**

A gas turbine engine's ignition system is a crucial component that provides the spark or flame required to start the combustion process in the engine's combustion chamber. The ignition system is in charge of igniting the fuel-air mixture at the proper moment and ensuring that the combustion process is sustained and controlled throughout the working range of the engine.

The ignition system is essential for assuring a gas turbine engine's safe and efficient operation. A faulty or failed ignition system can cause engine power loss, increased fuel consumption, and possibly catastrophic engine failure. As a result, gas turbine engines are built with redundant ignition systems and safety features to ensure that the engine can keep running securely even if the ignition system fails.

Spark Ignition
The Surface Discharge Igniter
Torch Igniter
Glow Plug
Plasma Jet
Laser Ignition
Chemical Ignition

**GLOW PLUG**

The function of a glow plug is to provide rapid reignition of the flame should extinction occur as a result of the sudden ingestion of water or ice, or through temporary fuel starvation. The dimensions of the plug are chosen to suit the size of the liner, but a typical plug would be in the form of a hollow cylinder, 25 mm in length and 15 mm in external diameter.

The main drawback to glow plugs is the obvious one—the risk of a plug, or part of a plug, becoming detached and damaging the turbine blades

The glow plug is mounted on the liner wall and protrudes into the primary zone where it is immersed in flame gases at high temperature. This is the ideal location for relighting the fuel–air mixture in the event of a sudden flameout

The function of a glow plug is to provide rapid reignition of the flame should extinction occur as a result of the sudden ingestion of water or ice, or through temporary fuel starvation. The dimensions of the plug are chosen to suit the size of the liner, but a typical plug would be in the form of a hollow cylinder, 25 mm in length and 15 mm in external diameter.

The main drawback to glow plugs is the obvious one—the risk of a plug, or part of a plug, becoming detached and damaging the turbine blades

The glow plug is mounted on the liner wall and protrudes into the primary zone where it is immersed in flame gases at high temperature. This is the ideal location for relighting the fuel–air mixture in the event of a sudden flameout

*Figure 4: Glow plug*

**SPARK PLUG**

A spark plug is a device for delivering electric current from an ignition system to the combustion chamber of a spark-ignition engine to ignite the compressed fuel/air mixture by an electric spark, while containing combustion pressure within the engine. A spark plug has a metal threaded shell, electrically isolated from a central electrode by a ceramic insulator. The central electrode, which may contain a resistor, is connected by a heavily insulated wire to the output terminal of an ignition coil or magneto. The spark plug's metal shell is screwed into the engine's cylinder head and thus electrically grounded. The central electrode protrudes through the porcelain insulator into the combustion chamber, forming one or more spark gaps between the inner end of the central electrode and usually one or more protuberances or structures attached to the inner end of the threaded shell and designated the side, earth, or ground electrode The distance between the tip of the spark plugs and the central electrode is called the "spark plug gap" and is a key factor in the function of a spark plug. Spark plug gaps for car engines are typically 0.6 to 1.8 mm (0.024 to

0.071 in) Modern engines (using solid-state ignition systems and electronic fuel injection) typically use larger gaps than older engines that use breaker point distributors and carburettors.


*Figure 5: Spark plug*

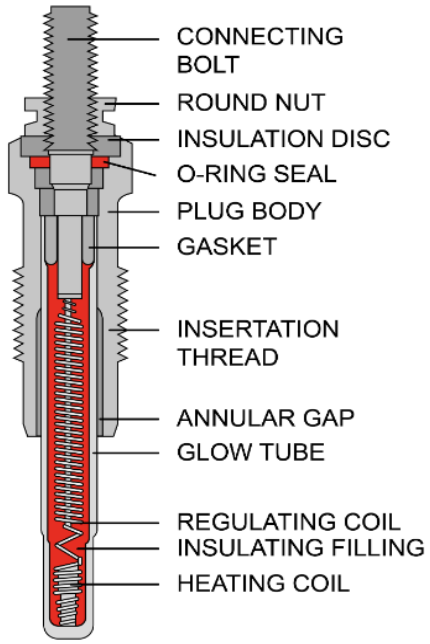



*Figure 6: Schematic Diagram of Glow plug*

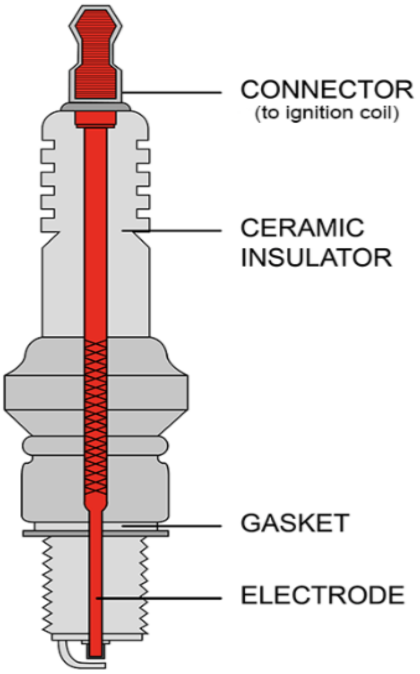



*Figure 7: Schematic diagram of spark plug*

**TORCH IGNITOR**

Torch ignitors are widely used in gas turbine engines to generate power, propel aircraft, and in other industrial applications. They are a dependable and efficient method of igniting the primary fuel and play an essential role in the safe and efficient operation of gas turbine engines. Torch igniter is made up of an additional fuel-air stream and a spark plug in a separate housing, where initial ignition is produced through the fuel mixture and spark plug, thus using the flame as an ignition source in the combustor by forming a torch.

The primary fuel is ignited by the formed torch.

Many variables influence torch igniter performance, including fuel volatility, air temperature, and air-fuel ratio.

Its position has no effect on its effectiveness.

It is typically installed in the annulus created between the liner and the air casing near the chamber's upstream end, but at least one annular combustor has been manufactured with a torch igniter mounted on the dome of the liner within the snout.

The main problem with torch ignitor is gasoline gumming and cracking.

The issue can be solved by installing solenoid valves that turn off the fuel after light up and by providing clean purging air, but these items add weight and complexity. The main problem with torch ignitor is gasoline gumming and cracking.

The issue can be solved by installing solenoid valves that turn off the fuel after light up and by providing clean purging air, but these items add weight and complexity.

Basically, torch ignition works on with certain steps like:

**Ignition sequence initiation**: A tiny amount of fuel is discharged by the torch ignitor nozzle of a gas turbine engine prior to the primary fuel being fed into the combustion chamber. Then, an electric spark or another ignition source ignites this gasoline.

**Flame stabilization**: The torch ignitor produces a modest, steady flame once it is lit. This flame acts as a constant source of ignition.

**Main combustion initiation:** The primary fuel supply is injected into the combustion chamber when the engine is prepared to start. The flame of the torch ignitor ignites the mixture created when the primary fuel and incoming air combine.

**Steady combustion**: The flame from the torch ignitor may continue to burn in addition to the main flame after the primary combustion process has started. This promotes stability and guarantees that the combustion process continues uninterrupted and dependably under all engine operating situations.

**Shut down and relight**: You may also utilise the torch ignitor for engine shut-down and restart processes. The torch ignitor can offer a dependable source of ignition to resume the combustion process if the main flame goes out for any cause.

Gas turbine engines must include torch ignitors, especially in circumstances when the main flame may fluctuate or extinguish, such as during startup, shutdown, and low-power operations. By supplying a reliable ignition source when required, they improve the combustion process' safety and dependability.

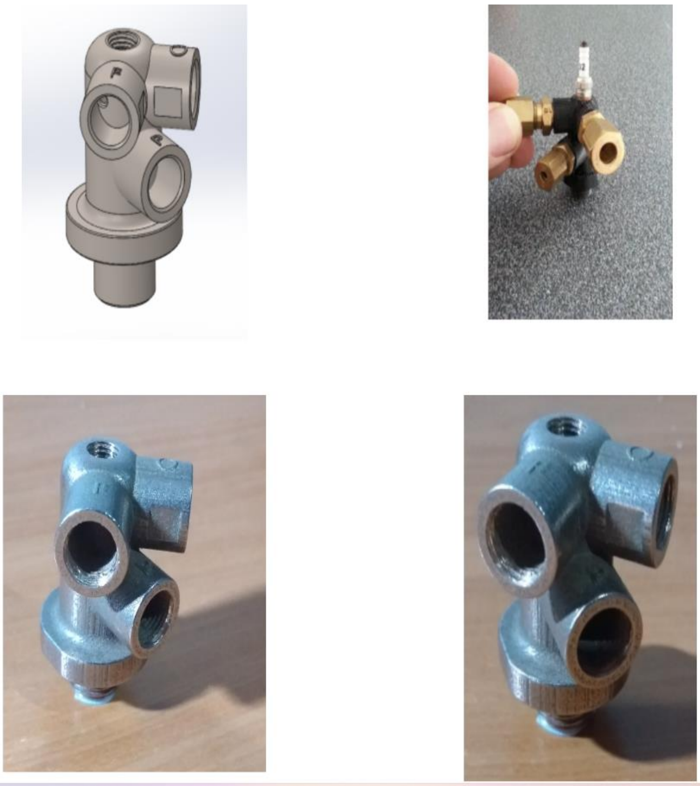


*Figure8: Torch ignitor model*

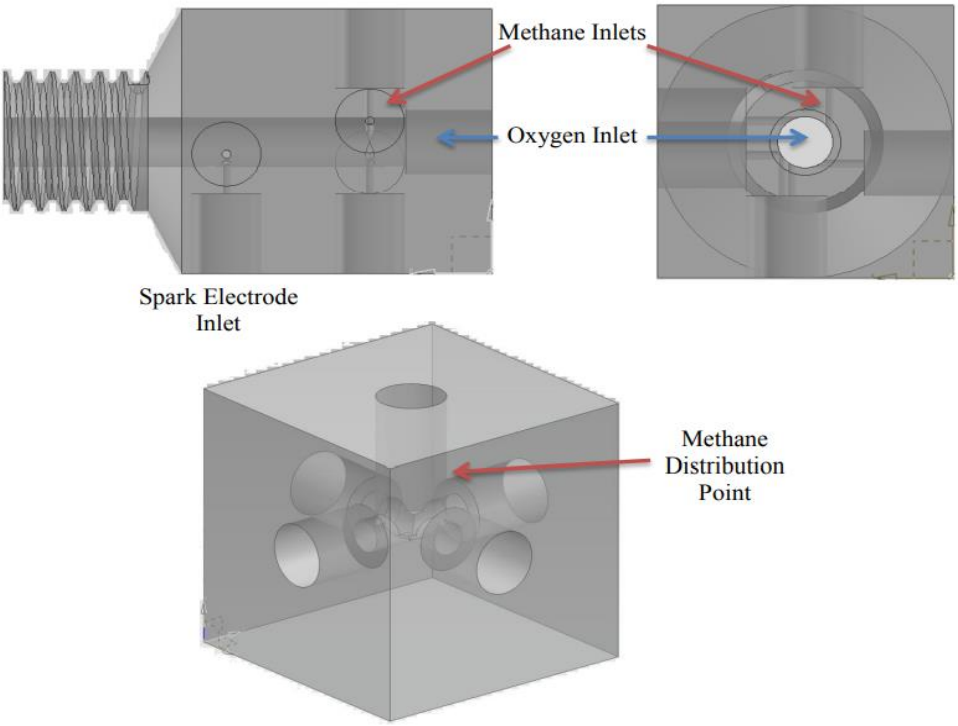



*Figure 9: Torch ignitor model for Rocket engine*

In gas turbine engines, the combustor section uses a torch ignitor, sometimes referred to as a pilot torch or igniter, to start the combustion process. In situations where a high-power engine is required, such as in aviation, power generation, and other fields, gas turbine engines are frequently utilised.

In order to ignite the fuel-air combination in the combustion chamber, the torch ignitor is a crucial component. It produces the initial flame. It guarantees consistent and dependable gasoline ignition, particularly at engine starter. Here's how it typically operates

**Ignition process initiation**: The torch ignitor is turned on when the gas turbine engine is started. It produces a small, controlled regulated flame that can ignite the fuel-air combination.

**Flame generation**: A spark plug or other similar device is frequently used by the torch ignitor to generate a regulated spark or flame. The primary fuel-air combination in the combustion chamber is ignited by this flame.

**Flame propagation**: The primary fuel-air combination in the combustion chamber is ignited by the first flame produced by the torch ignitor. By starting the combustion process, energy is released, and high-temperature, high-pressure gases are produced.

**Combustion and power generation**: As the ignited fuel-air mixture burns quickly and produces a high-speed flow of hot gases, the combustion process continues. The turbines of the engine are made to spin by the passage of these gases through the turbine section. The compressor and, occasionally, external loads like the aircraft's propulsion system or a power generator, are linked to the rotating turbines. Depending on the use, this rotation produces propulsion or mechanical power.

When the engine needs to start up from a stop or at high elevations where the air is thin, the torch ignitor is essential for ensuring that the combustion process starts reliably and rapidly. The combustion process wouldn't start without adequate ignition, and the engine wouldn't be able to generate the necessary power. The basic idea of employing a torch ignitor to start combustion is constant across most designs, however different gas turbine engines could use somewhat different designs and technology for their ignition systems. The effectiveness, dependability, and performance of gas turbine engines continue to be enhanced through developments in ignition technology.

## CANDIDATE DESIGNS FOR TORCH IGNITOR

### ULTRA COMPACT COMBUSTORS

A form of combustion chamber that is smaller and more efficient than conventional combustion chambers is known as an ultra-compact combustion chamber. These chambers are most commonly found in small engines, such as those found in drones or small generators, where room is limited and weight reduction is essential.

An ultra-compact combustion chamber is typically designed using advanced materials and aerodynamic shaping to optimize combustion efficiency while reducing overall chamber size.Overall, ultra-compact combustion chambers are an important technology for improving the efficiency and environmental performance of small engines, especially in applications with limited space and weight.

To achieve high combustion efficiency and low emissions, ultra-compact combustion chamber designs frequently integrate cutting-edge technologies like micro-mixing, lean premixed combustion, and flame stabilization techniques. Faster reaction times, quicker start-up and shut-down, and lessened weight and space requirements are all made possible by the combustion chamber's small size In general, ultra-compact combustion chambers are a significant technological advancement for applications involving small-scale power production, where weight, size, and efficiency are important considerations.

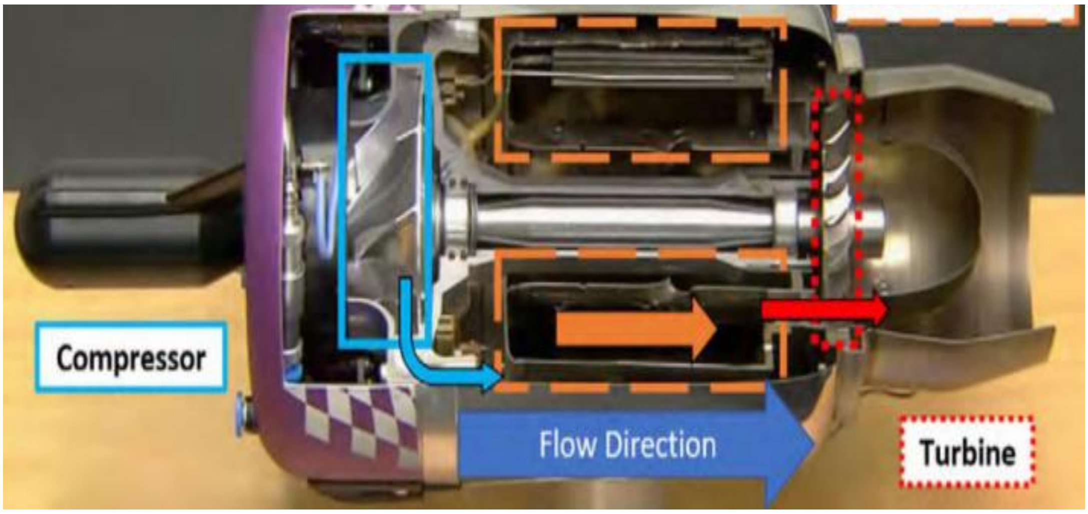


Figure 10: Ultra compact combustors

**VORTEX COMBUSTORS**

TVC consists of three main components, a forebody, cavity, and an afterbody. Bulk flow travels past the forebody creating a shear layer at the edge of the cavity.

Gas turbine engines and other combustion systems use trapped vortex combustors as a form of combustion chamber. They are made to increase steadiness, lower emissions, and improve combustion efficiency.

Air driver jets from the afterbody inject air into the cavity opposing the direction of the bulk flow. The air driver injection coupled with the shear layer establishes a trapped vortex which aids as a flame holder to promote stable combustion.

This vortex allows for a rapid mixing and exchange of products between the cavity and bulk flow.

**HIGH G COMBUSTORS**

Utilizes high-g centrifugal loading as its mechanism to shorten its length while ensuring sufficient residence time. To introduce g-loading into the cavity, air is introduced circumferentially.

This g-loading forces the heavier unburned gases to the outer diameter of the cavity while the lighter products migrate toward the bulk flow to be entrained and exhausted.

This radial migration significantly increases the residence time since the unburned reactants are held within the cavity until they are burned

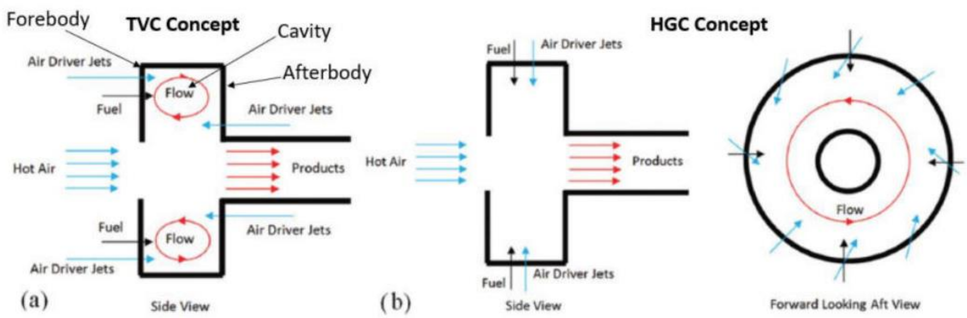



*Figure 11: TVC and HGC*

**LIQUID FILIM COMBUSTORS**

Liquid film combustors are a form of combustion system used in aerospace and gas turbine applications. They are also known as wetted-wall combustors or thin-film combustors. They work by introducing a thin liquid film onto the interior surface of the combustion chamber, which serves to cool the walls while also improving fuel-air mixing.

The liquid fuel in a liquid film combustor is usually atomized and sprayed onto the combustion chamber wall, where it forms a thin film. The combustion chamber is then filled with air or another oxidizer, which mixes with the fuel film and ignites, producing a high-temperature, high-pressure gas that can be used for propulsion or energy production.

The ability of liquid film combustors to work at high power densities is one of their main benefits, making them ideal for applications where space and weight are limited. Furthermore, the use of a liquid film can aid in the reduction of harmful pollutants such as nitrogen oxides and carbon monoxide.

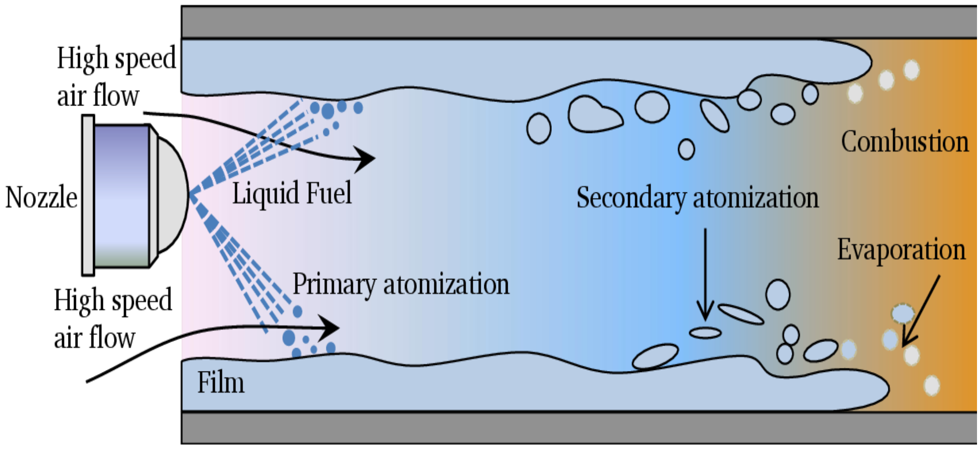



*Figure 12: Liquid film combustors*

**ATOMIZATION**

It involves breaking down the liquid fuel into tiny droplets or particles, creating a fine mist that can mix more effectively with air before being ignited.

This process is essential for efficient and clean combustion, as it maximizes the surface area of the fuel exposed to the air, allowing for better mixing and more complete combustion.

Selection of fuel nozzles which provides better breakdown of fuels into tiny droplets is an important aspect for torch ignitor development phase

**Fuel nozzle**

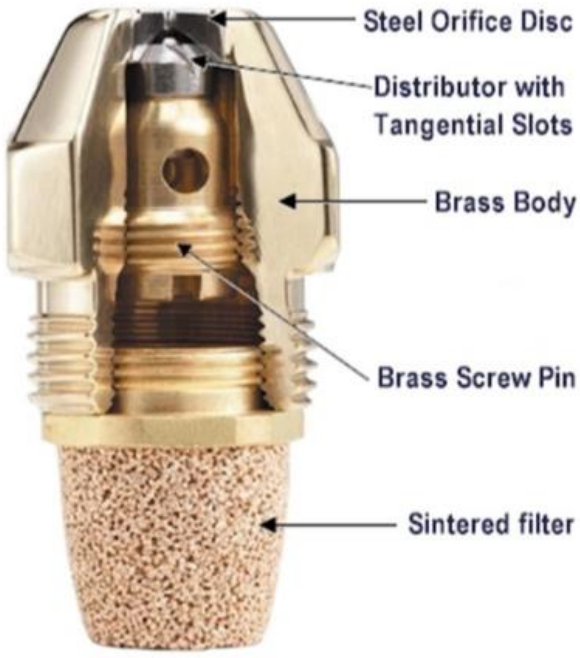


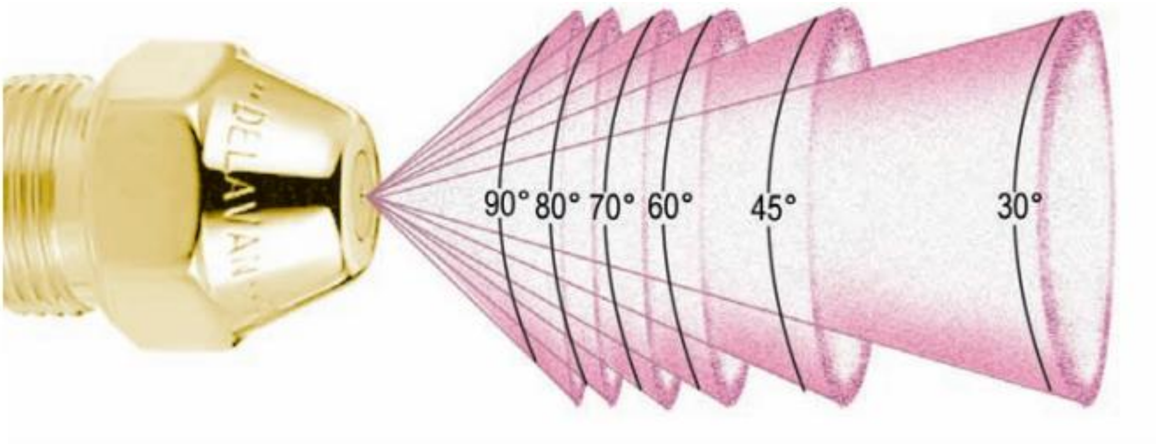

*Figure 13: Fuel nozzle*

There are several criteria to select the fuel nozzle, among which the important parameters are:

1. **Gas oil flow rate**
2. **Gas oil spray angle**

### 3. Gasoline spray form

**Oil spray angle (angle determined on each nozzle)**




The angle at which the cone of spray emerges from the nozzle is referred to as spray angle. Available spray angles to accommodate the vast range of burner air patterns and chamber forms, from a 30-degree angle to a 90-degree angle. Generally, round or square chambers are fired with 70 to 90-degree nozzles. Short wide chambers need a short fat flame. Long narrow chambers usually require 30-degree to 70-degree solid cone nozzles The spray pattern and angle must be such that every drop burns entirely in suspension in the combustion area. The correct spray pattern and angle depends on the air-oil mixing design of the burner and the shape of the combustion chamber

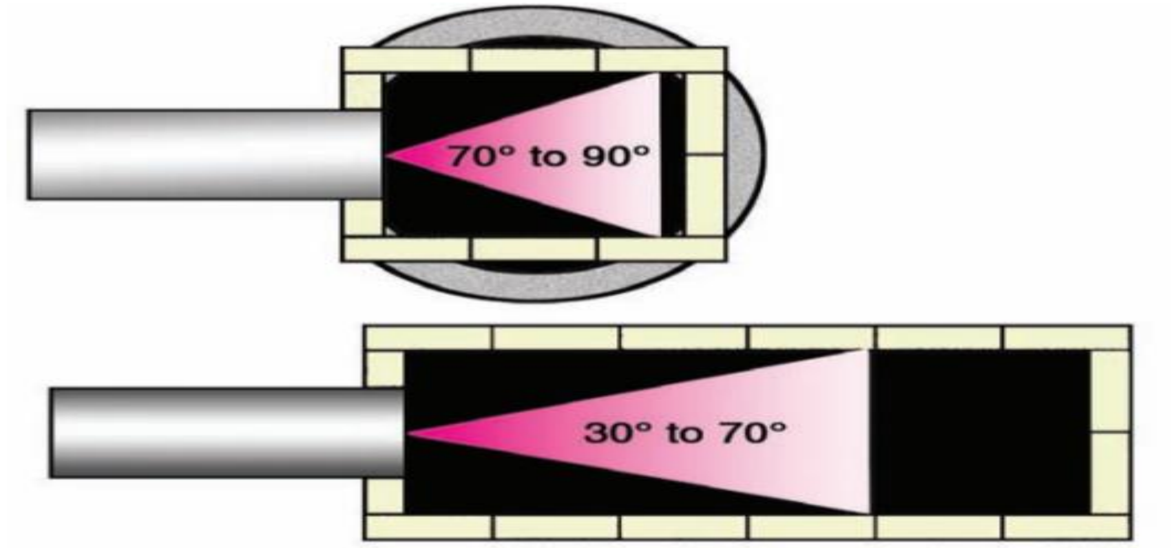



*Figure 15: Spray angle inside chamber*

**Spray form (letters B, H, S, ... specified on each nozzle)**

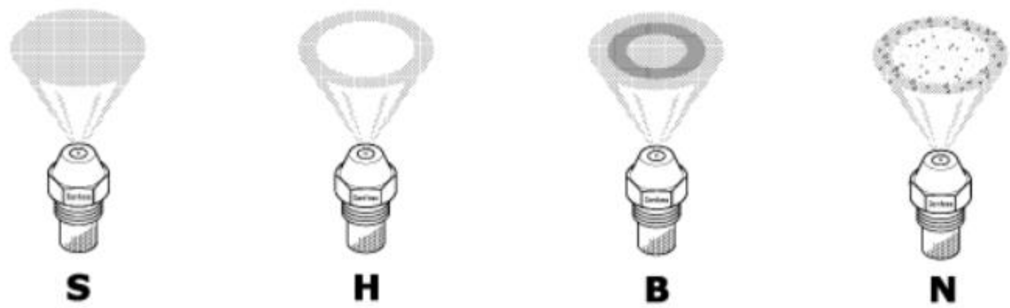



*Figure 16: Spray forms*

**Table of Oil Burner Nozzle Patterns & Properties**

| Oil Burner Nozzle Selection, Patterns & Properties | |
|---|---|
| **Oil Burner Nozzle Spray Characteristics** | **Nozzle Spray Pattern Code** |
| Hollow | A, C, H, NS |
| Solid | B, P, R, S |
| High Velocity | HV |
| Semi Hollow | CM, SH |
| Semi Solid | PLP, SS |
| Special Design | Q |
| Special Design | W |
| Special Design - anti rumble | AR |
| Hollow or Solid | H or S |
| Extra Solid | ES |
| Extra Hollow | EH |

*Figure 17: Nozzle Patterns*

## CHAPTER 2
## COMPUTATIONAL FLUID DYNAMICS

CFD is a branch of science primarily concerned with material analysis related to the fluid flow, heat transfer and related phenomena such as mixing and combination through computer simulation. This process is used in a variety of industrial and non-industrial application. Despite being a relatively new and evolving field of science, CFD has received significant attention around the world for its diverse applications. The advantage of CFD include the ability to model complex problems that are difficult, time consuming and expensive to study by using experimental technique and analytical methods. Today's aerospace industry is highly diversified, encompassing a plethora of military, commercial and industrial applications. Computational analysis is critical for aerospace application and offers a cost effective and sophisticated solution in an expensive market. Although the reliability of CFD tools has historically been debatable, miscalculations, eliminating the errors.

All CFD simulation are approached by following the steps outlined below.
*Define modelling goals
*Identify the area of interest
*Creation of a solid model of the domain
*Generate the mesh
*Calculate the solution
*Examine the results

Pre-Processor
Pre-processing consists of inputting a flow problem into a CFD program through a user-friendly interface, and then converting the input into a form suitable for the solver. User activities in the pre-processing phase include
*Defining the geometry of the region of interest: the computational domain
*Grid generation: the subdivision of the domain into several smaller, non-overlapping subregion: a grid of cells
*Selection of physical and chemical phenomena to be modelled
*Definition of the properties of fluids
*Indicate the corresponding boundary conditions in the cells that coincide with or touch the boundary of the domain.

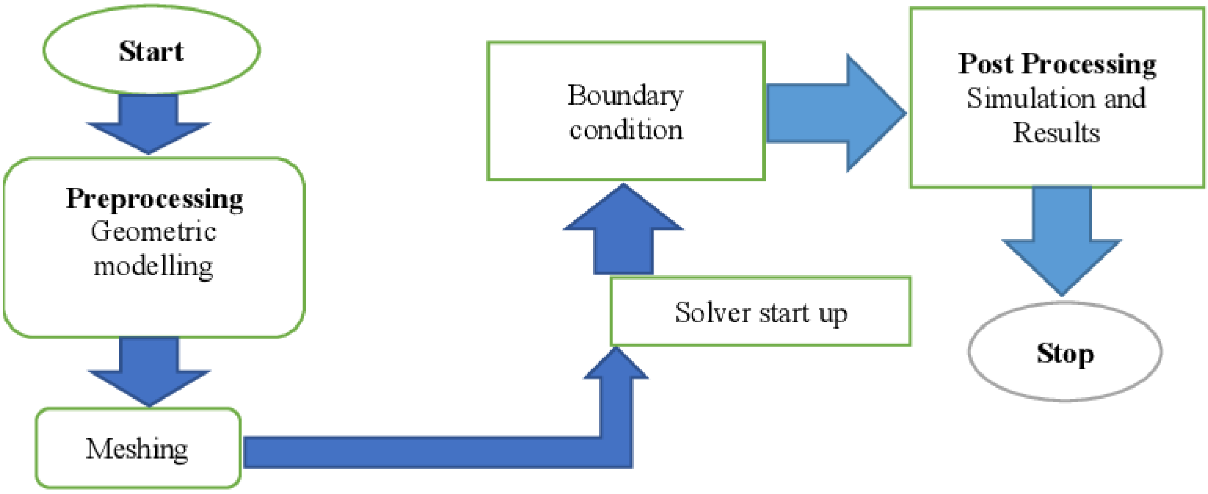



*Figure 18: Steps involved in CFD*

**Solver**

There are four different methods of numerical solution techniques. Finite difference, finite element, spectral and finite volume methods. Numerical methods perform the following steps:
*Approximation of unknown flow variables using simple functions.
*Substitution discretization of the approximations in the relevant flow equations and subsequent mathematical manipulations.
*Solution of algebraic equations

**Discretization Technique**

The finite volume method (FVM) is used to analyse the internal flow problem for discretization. This technique first device the entire domain into smaller section known as control volumes. Then the different equation governing fix are integrated over each volume to calculate the variables of interest at the cell centroid. FVM can also be used for unstructured meshes, which offers an advantage over the finite difference method (FDM). Non uniform coarse grid also given accurate solution when compute using this technique. An advantage of the FVM over finite difference method that it provides an accurate solution for both unstructured meshes and coarse, non-uniform meshes. In addition, FVM is the ideal method to resolve discontinuity that occur in a compressible flow. The finite difference method describes the unknown problem by point samples at the nodes of a grid or coordinate line. Finite element method uses simple piecewise function (e.g., linear or quadratic) valid on to describe local variation of unknown flow variables

**ANSYS fluent**

Fluent is an advanced computational tool for modelling fluid flow and heat transfer problems for an extended range of generations. The software provides modelling capabilities for incompressible and compressible, as well as laminar and turbulent flow problems. Fluent has the capability to solve problems having unstructured, apart from the structured mesh. This reduces computational time to users while grid generation and simplification of the geometry. Furthermore, the flow fields with large gradients can estimated with accuracy in the solution adaptive grid capabilities. This feature also reduces the computational time required to achieve the desired level of accuracy. All the utilities required to compute a solution and present the results are accessible in the softball by means of an interactive, menu- driven interface. One of the central components of fluent tool is Fluent Meshing, which was used in the current study for the geometric modelling and grid generation of the Domain.

**Basic Governing Equation**

Navier-Stokes equations are the governing equations of Computational Fluid Dynamics. It is based on the conservation law of physical properties of fluid. The principle of conservational law is the change of properties

- Continuity Equation
- Momentum Equation
- Energy Equation

Coordinates: (x,y,z)
Velocity Components: (u,v,w)
Time : t
Density: ρ
Total Energy: Et
Pressure: p
Stress: τ
Heat Flux: q
Reynolds Number: Re
Prandtl Number: Pr

**Continuity:**

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

**X – Momentum:**

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{xz}}{\partial z}\right]$$

**Y – Momentum:**

$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}=-\frac{\partial p}{\partial y}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{yz}}{\partial z}\right]$$

**Z – Momentum**

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z}=-\frac{\partial p}{\partial z}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}\right]$$

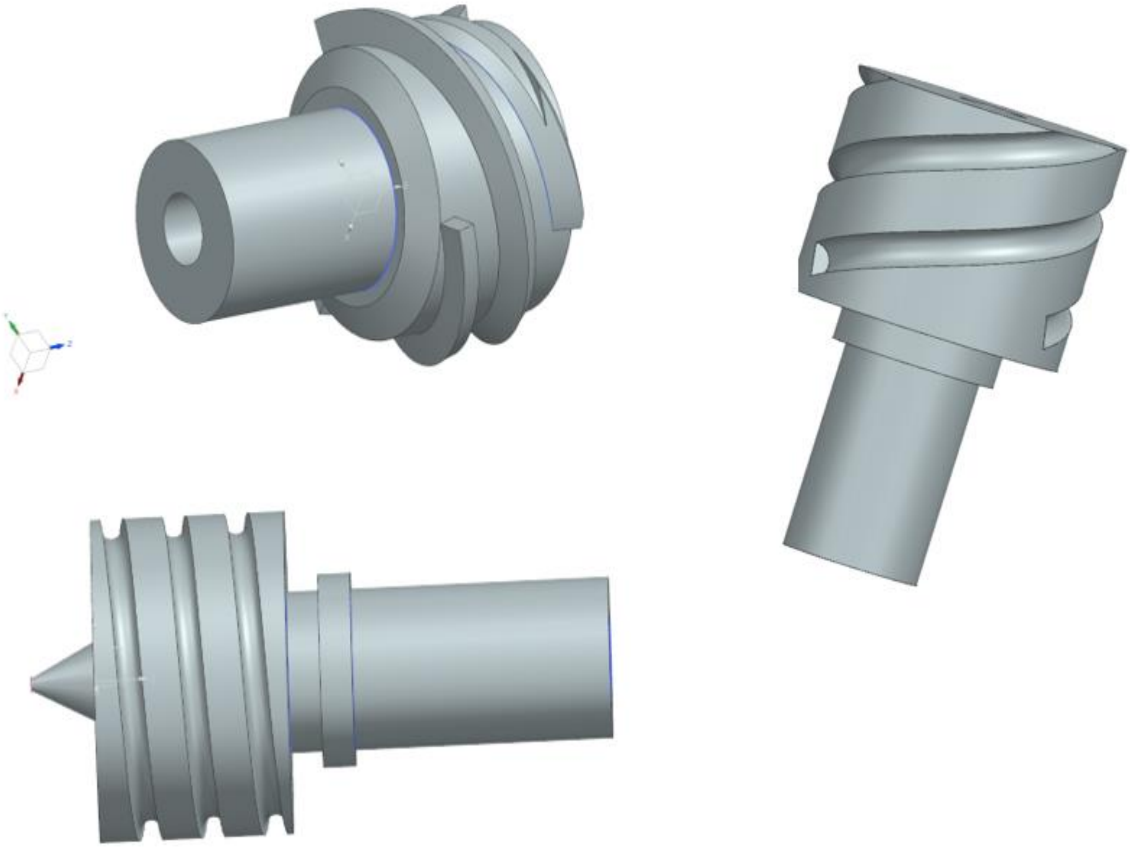
**Energy:**

$$\frac{\partial(E_T)}{\partial t}+\frac{\partial(uE_T)}{\partial x}+\frac{\partial(vE_T)}{\partial y}+\frac{\partial(wE_T)}{\partial z}=-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}-\frac{1}{Re_r Pr_r}\left[\frac{\partial q_x}{\partial x}+\frac{\partial q_y}{\partial y}+\frac{\partial q_z}{\partial z}\right]$$
$$+\frac{1}{Re_r}\left[\frac{\partial}{\partial x}(u\tau_{xx}+v\tau_{xy}+w\tau_{xz})+\frac{\partial}{\partial y}(u\tau_{xy}+v\tau_{yy}+w\tau_{yz})+\frac{\partial}{\partial z}(u\tau_{xz}+v\tau_{yz}+w\tau_{zz})\right]$$

**Post processing**
After the computational analysis is finished, post-processing in computational fluid dynamics (CFD) entails the analysis and visualisation of simulation findings. Post-processing makes it possible to visualise the intricate flow physics and improves comprehension of the simulation findings.
Following are a few typical methods for CFD post-processing:
* Contour plots are 2D or 3D diagrams that display how a particular parameter—such as velocity, pressure, or temperature—varies over a predetermined area of the modelling domain.
* Streamlines are lines that show the path of fluid flow at a specific moment in time. Using streamlines, it is possible to see the flow patterns, spot regions of recirculation, and locate the separation points.
* Velocity vectors are arrows that depict the magnitude and direction of the velocity at various locations throughout the simulation area. You can see the flow direction and strength using velocity vectors.
* Histograms are plots that display the frequency distribution of a specific quantity, like pressure or velocity.
* Animation: Animations are frequently used to show how the flow field changes over time. Animations can be used to pinpoint the beginning of turbulence or the position of vortices, as well as to demonstrate how flow patterns change over time.

## CHAPTER 3
## MODELLING AND VALIDATION STUDY

The CAD geometry of the torch ignitor used for the analysis is shown in fig xx, designed using Unigraphics NX or NX, which is a high-end multi-purpose CAD/CAM/CAE programme that is used for parametric and direct solid or surface modelling. It is also employed in engineering analysis, where the finite element and finite volume approaches are applied.
Computer-aided design, manufacturing, and engineering (CAD/CAM/CAE) software Siemens NX is a sophisticated and all-inclusive programme created by Siemens PLM Software. NX has been a popular option for engineers, designers, and manufacturers across a range of industries thanks to its lengthy history and solid reputation in the field.
The core of NX is its robust 3D modelling functionality, which enables users to generate elaborate and realistic designs of products and components. Design changes can be made quickly and easily with the help of the programme, which enables feature-based and parametric modelling. Additionally, NX provides cutting-edge surface modelling capabilities that are essential for creating complicated freeform forms and Class-A surfaces for the aerospace and automotive sectors.
The Torch ignitor was designed from the initial stage after many literature reviews and the appropriate design was opted after considering certain factors like swirling mechanism, pressure drop, flame length etc.



With the help of certain literature reviews, it was found that the torch ignitor model discussed in the paper ‘’Experimental and Numerical Study of the Flammability Limits in a CH4/O2 Torch Ignition System” by Olexiy Shynkarenko, Domeico Simone, Jungpyo Lee, Artur Bertoldi and series of papers discussed with the same geometric model was apt for the work of Torch ignitor for gas turbine engine Three geometrical features for swirling mechanism were selected, out of which one which gives almost constant pressure throughout the chamber during cold flow analysis was taken for further analysis purpose.

From the geometrical models in fig 19, we have selected model no. 2 because it shows minimum pressure loss.
There are three helical slots within the geometry at 120-degree angle between each swirling slot and diameter of helix is 40mm, with pitch of 35.5mm and 0.7 turn.
The swirling section of the body is with diameter of 40mm and 25mm in length, the overall length of the swirling body is 62mm.
The air flow pass through the helical slot and create a swirling motion, the swirled air is then mixed with fuel at appropriate air to fuel ratio.

*Figure 19: Models for swirling effect*

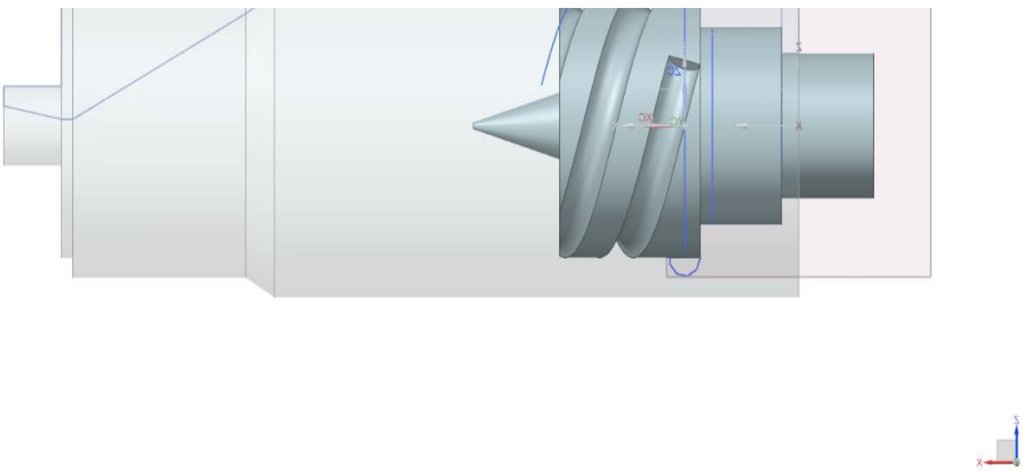


*Figure 20: Swirl body within the casing*

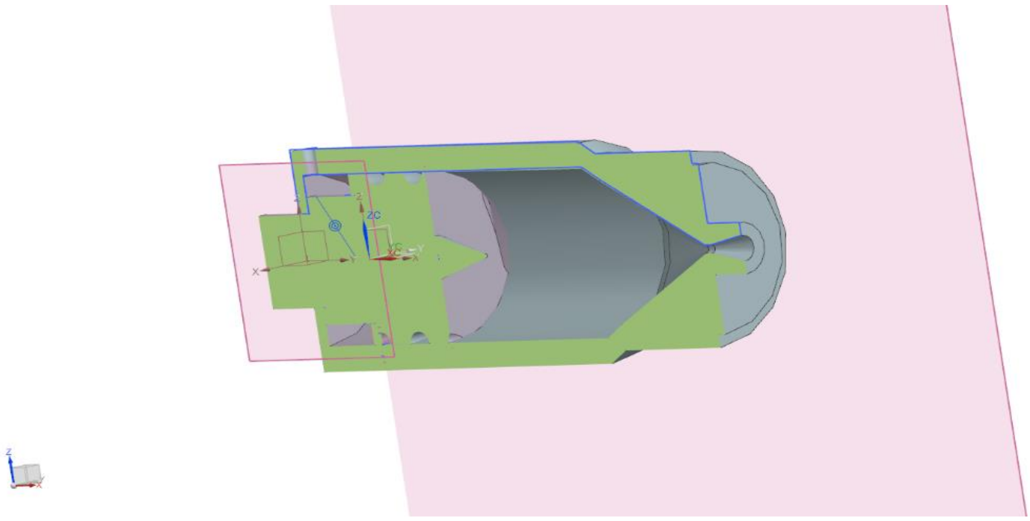


*Figure 21: Cut section of casing*

Fuel flows inside the swirling body, but the fuel flow doesn't follow the swirling mechanism and it escapes to the chamber through the conical nozzle outlet with diameter of 1mm

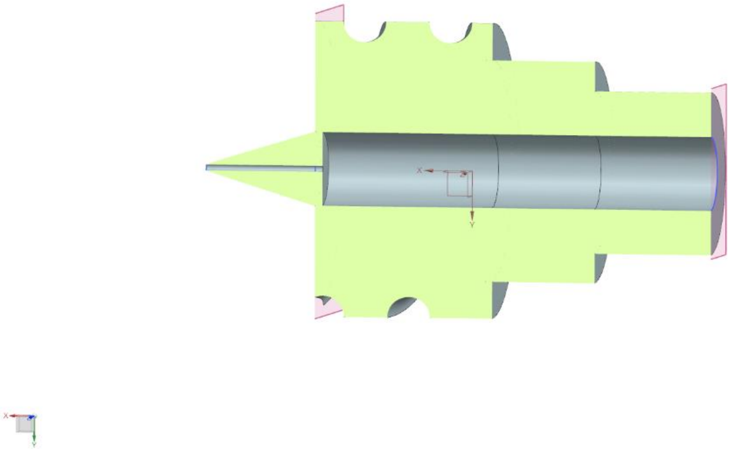
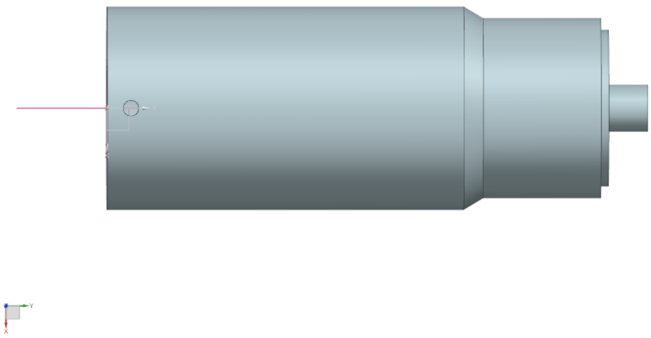


Figure 22: Cut section of swirl body

Swirling body is kept inside a casing in which the combustion takes place, the casing is of 138mm where 52mm of the swirling body fits inside the casing/ chamber.
Air flows inside the chamber through an inlet of 4mm diameter located at a distance of 6mm from the casing end
The casing is of constant diameter of 40mm till 91mm of its length and then it converges to a diameter of 2mm for the next 35mm of its length, the throat diameter is 2mm and extends to a length of 2mm, then the casing body diverges to a diameter of 6mm within a length of 10mm.



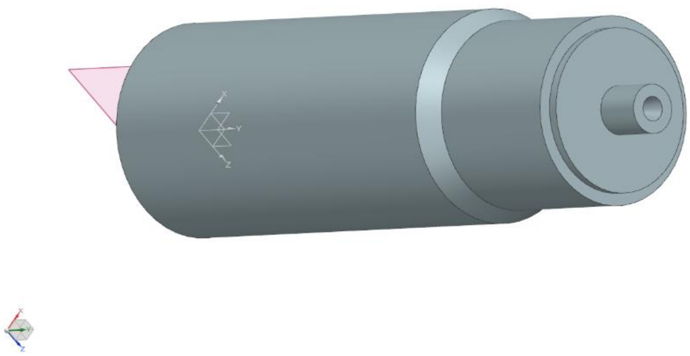


Figure 23: Torch ignitor case view 1

Figure 24: Torch ignitor case view 2

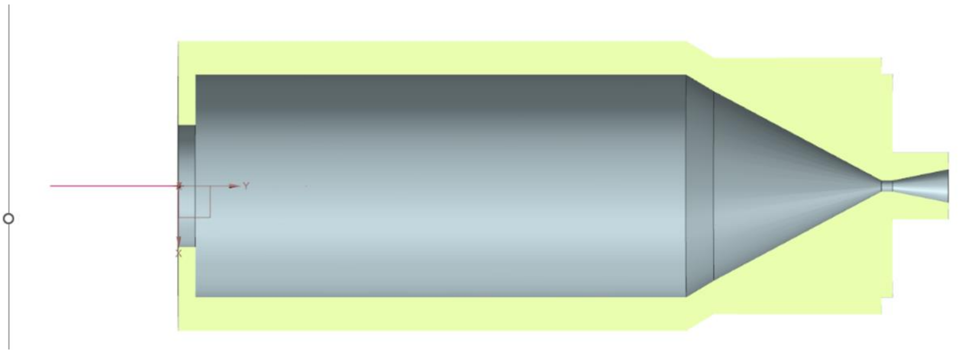
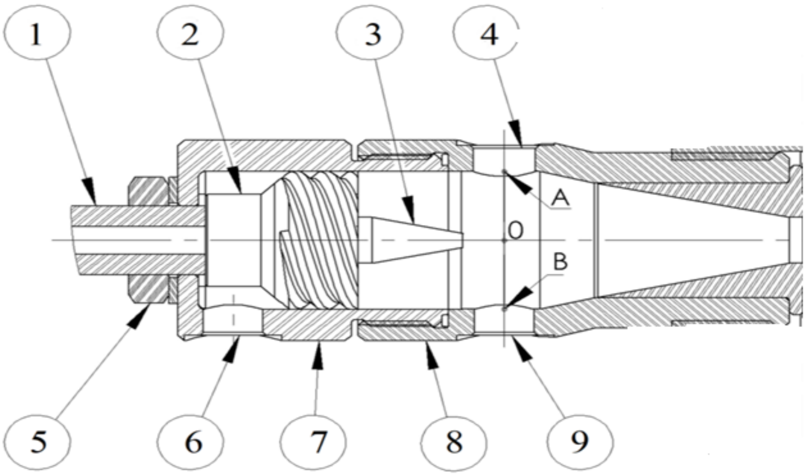


*Figure 25: Torch ignitor case view 3*




| | |
|---|---|
| 1 | Fuel inlet |
| 2 | Air injector |
| 3 | Fuel injector |
| 4 | Sensor interface |
| 5 | Fuel inlet fastener |
| 6 | Air inlet |
| 7 | Igniter casing |
| 8 | Combustion chamber, |
| 9 | Spark plug interface |

Fuel is injected by means of a jet parallel to the axis, air enters the combustion chamber passing through helical ducts which impart the
desired circumferential motion.
The swirled flow thus, surrounding the fuel jet, controls the extension
of the reacting region and its temperature, by increasing or decreasing the oxygen mass flow rate it is thus possible to protect the walls until the exit section. Downstream of the fuel injector are located the
spark plug and thermocouple interface

**SPACE CLAIM**

The geometry has been modelled in CAD software. Fluid extraction and geometry cleaning was performed in space claim. Once the fluid extraction is done, the solid part of the geometry is
Circular domain was created using space claim where the length of domain is 100mm and the diameter of circular domain was 60mm, domain is created in order to visualise how the flow develops in the atmosphere.
Throat and divergent portion of torch ignitor is placed within the domain, two body of influence is created inside the domain which is used only for meshing purpose. Naming of components are done in space claim which makes it easier to mesh the geometry individually and also to apply specified boundary conditions during analysis

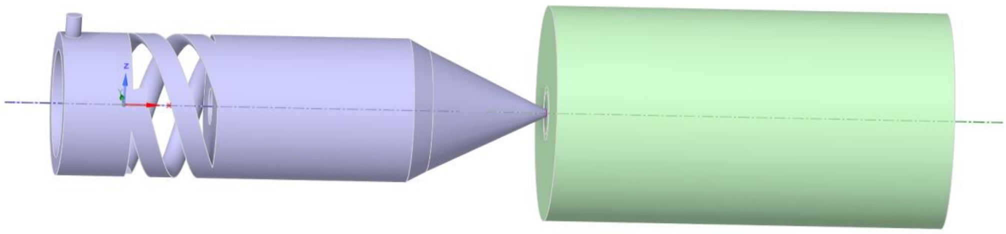


*Figure 26: Ignitor body with domain*

body of influence (BOI) is created within the geometry in order to have finer mesh in those specified area. This region is only used for meshing purpose to have separate mesh. In this geometry there are two boi created, one with 12mm dia. and 100mm in length and the second one with 30mm dia. and 80mm in length, this will help us to visualise the exit flow from the torch ignitor

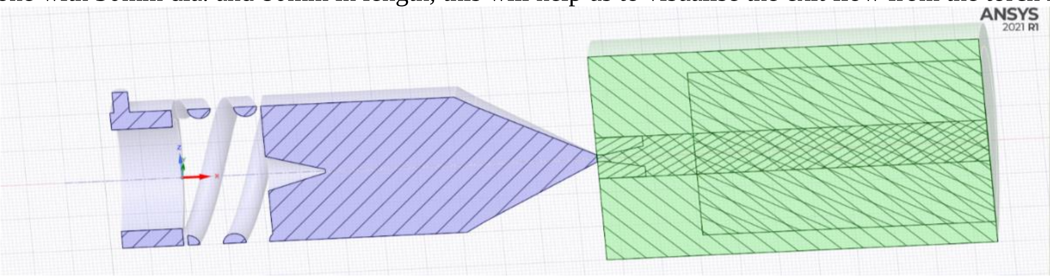



*Figure 27: Cut section of fluid extracted body*

## MESHING

In order to increase simulation precision and perform finite element analysis, meshing is a technique used in Ansys FLUENT. Grids in two and three dimensions are used in the mesh generation procedure. According to our sizing, it broke down complex geometries into small components to discretize the domain. It enables the design to produce real-time, precise mesh models. It uses a numerical grid to apply to the border and geometry. If the mesh is bigger, the simulation lasts longer; if the mesh is smaller, the simulation lasts shorter and the solver times lengthen. Prior to that, import the model into Space Claim to build a domain with 10D and 30D length. Then, construct a BOI to improve mesh with different diameter and extracted volume for all models.

### Types of Mesh

### Structured mesh

Structured meshing, also known as mapped meshing derives its name from the fact that the mesh points define an IxJ array of quadrilaterals (in 2D) and an IxJxK array hexahedra (in 3D). The structure is implicit because, at any mesh point, its neighbours are implicitly known. This structure adds efficiency to the mesh generation and flow solver algorithms. A structured mesh's structure makes it difficult to create for complex shapes, and in practise, it usually employs a multi-zone process in which numerous structured grids are stitched together. CFD solutions computed on quad and hex structured grids are thought to be more accurate than other cell types.

### Unstructured meshing

Unstructured meshing is commonly used to describe meshes made up of triangles (in 2D) and tetrahedra (3D). Their unstructured nature (in comparison to structured meshes) stems from the fact that the neighbours of any grid point must be explicitly established using some type of look-up. The Delaunay criteria or an advancing front technique is commonly employed in the generation of unstructured meshes. Geometrically complicated shapes can be meshed with relative ease using these methods. However, due to the lack of alignment of the grid lines, the accuracy of CFD simulations performed using pure unstructured meshes can be worse than that of structured grids, particularly in places such as the near-wall boundary layer.

### Hybrid Meshing

Most recent CFD simulations utilise a hybrid mesh that combines cell types from structured and unstructured approaches (hexahedra and tetrahedra) with prisms and pyramids to achieve the best of both worlds (accuracy and geometric flexibility). More notably, the near-wall mesh employs a semi-structured approach to resolve the boundary layer, transitioning to various cell types as the mesh travels away from the geometric mode.

## GRID DETAILS AND GRID INDEPENDENCE STUDY

The geometry was meshed using Ansys Fluent meshing software with watertight geometry directly imported from Ansys Space claim

The geometry was meshed such that the walls of the torch ignitor was having finer mesh compared to the central region of the ignitor, also the ignitor body was meshed in fine size than the computational domain

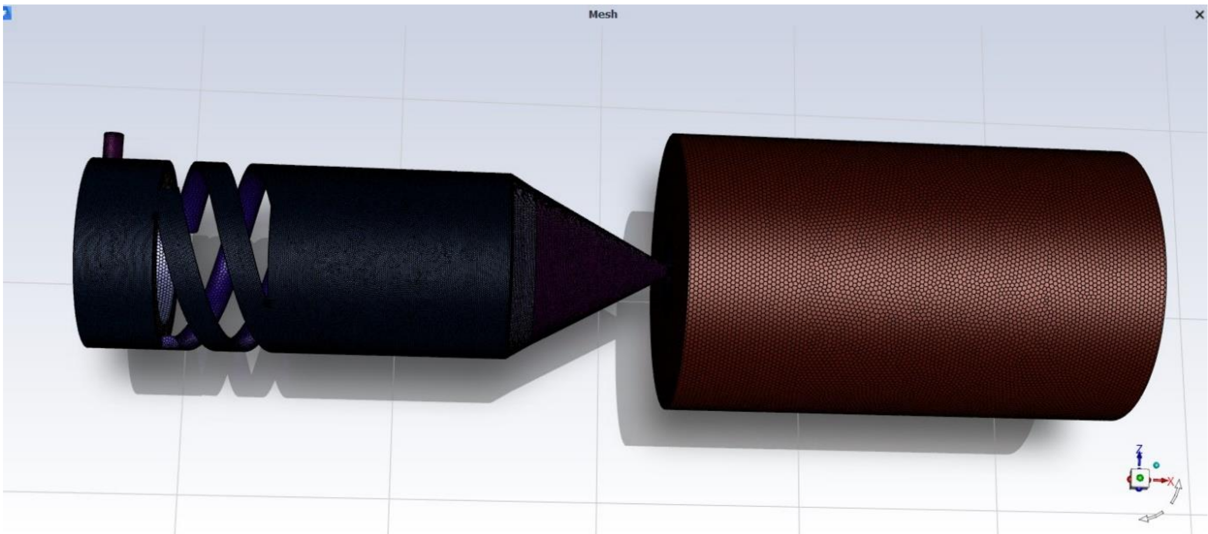
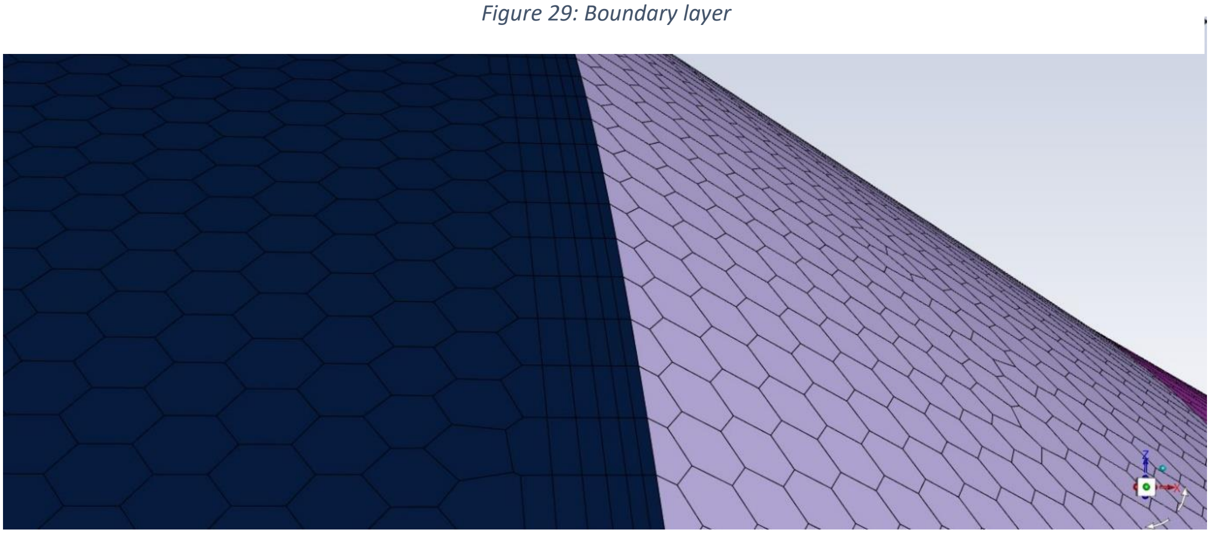



*Figure 28: poly hexcore mesh*

Both surface meshing and volume mesh were done using Ansys Fluent, volume meshing was done using Poly -hexcore mesh with 5 boundary layers

*Figure 29: Boundary layer*



**GRID INDEPENDENCE STUDY**

As part of the grid independent study, simulations were run with various grid sizes and elements to test the sensitivity of the mesh. Surface and volume mesh were created using Ansys fluent software

The grid independent study for the geometry with the given boundary conditions of pressure inlet and pressure outlet with Reynolds-averaged Navier strokes (RANS) turbulence models of Realizable k-Ɛ with standard wall treatment.

The grid independent study will help to find the minimum cell count with the minor change of the analysis performance

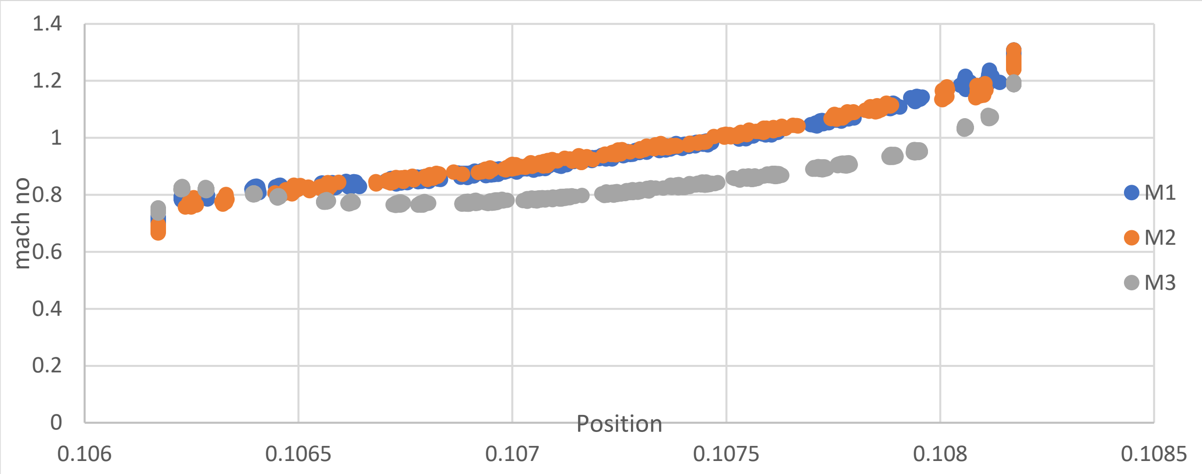



*Figure 30: Grid independent study*

| GRID NAME | M1 | M2 | M3 |
|---|---|---|---|
| No. of cells | 4.29M | 2.19M | 0.93M |
| Maximum Skewness | 0.799486 | 0.799696 | 0.799435 |
| Minimum Orthogonality | 0.20 | 0.20 | 0.20 |

*Table 1: Grid Details*

| | |
|---|---|
| Energy | On |
| Viscous | K-epsilon |
| | Realizable |
| | Standard wall function |
| Species | Non-Premixed |
| | Steady diffusion flamlet |
| DPM | On |

From grid independent study on torch ignitor with pressure as the boundary conditions, it is found out 2.19M grid because grid sensitivity is less very less and it won't affect the Mach Number. Mach Number was plotted for all the grids after performing numerical analysis with the mentioned boundary conditions.
After fluid extraction and geometry cleaning from Ansys spaceclaim, it is directly imported for meshing. The geometry was divided into two parts as torch ignitor part and domain part
Meshing was done using Ansys fluent meshing with work flow option as Watertight geometry, where all parts are meshed with different size according to their importance in the flow analysis and size of the geometry using add local sizing option
Surface meshing was done with curvature and proximity as size function with minimum cell size of 0.15mm and max cell size of 2.4mm with growth rate of 1.2 and curvature normal angle of 18
The geometry consists of only fluid region with no void, also boundary types were mentioned before volume meshing process and then boundary layers were added to the geometry with smooth transition with 5 layers
Volume mesh was done using Poly- hexcore with minimum and maximum cell length of 0.15 and 2.4 respectively
After meshing, the overall cell count was 2194538 where domain was having 1480393 cells with 0.799 skewness and the ignitor body was with 714145 cells and 0.797 skewness
The mesh was with an overall skewness of 0.7996 with orthogonal quality of 0.2
*Table 2: CFD Solver input*

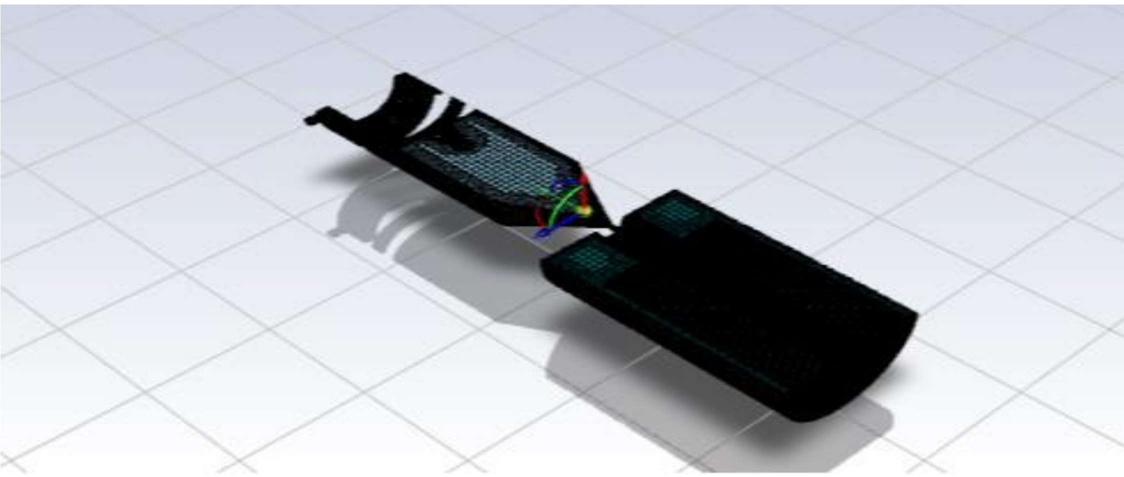


*Figure 31: Cut section of mesh*

**CFD SOLVER - FLUENT**

After meshing the geometry, it was then imported to Ansys Fluent for analysis where all the required parameters like turbulence effect, cell zoned condition, boundary conditions, material specification, solution controls etc are specified.

**Energy equation**
The Navier-Stokes energy equation follows the energy conservation law, which correlates a system's total energy to the sum of work and heat added to the system. The Navier-Stokes energy equation is used in CFD simulations to explain the energy associated with flow behaviour

$$\underbrace{\rho\left[\frac{\partial h}{\partial t}\right.}_{\mathrm{I}} + \underbrace{\left.\nabla\cdot(h\vec{V})\right]}_{\mathrm{II}} = \underbrace{-\frac{Dp}{Dt}}_{\mathrm{III}} + \underbrace{\nabla\cdot(k\,\nabla T)}_{\mathrm{IV}} + \underbrace{\phi}_{\mathrm{V}}$$

The energy equation stated above is one way to express the

Navier-Stokes equations for compressible flow. In the case of ideal fluid flow, the Navier-Stokes energy equation can be reduced further in terms of enthalpy.

$$\rho c_p \left[\frac{\partial T}{\partial t} + (\vec{V} \cdot \nabla)T\right] = k\nabla^2 T + \phi$$

**Turbulence effect**

Turbulence modelling is the construction and use of mathematical model to predict the effects of turbulence. The equations governing turbulent flows can only be solved directly from simple cases of flow. For most real-life turbulent flows, CFD simulations use turbulent models to predict the evolution of turbulence. These turbulence models are simplified constitutive equations that predict the statistical evolution of turbulent flow.

Most commonly used turbulence models are k-ε, k-ω, k-ω SST, Spalart-Allmaras etc.

For our case we have taken turbulence model as k-ε realizable standard wall function

k-ε: It is a two-equation model that gives a general description of turbulence by means of two transport equations (PDEs). The original impetus for the k-ε model was to improve the mixing length model, as well as to find an alternative to algebraically prescribing turbulent length scale in moderate high complexity flows. The two transport variables are turbulent kinetic energy k, which determines the energy in the turbulence, and the turbulent dissipation (ε), which determines the rate of dissipation of the turbulent kinetic energy

The k- epsilon model predicts well away from the boundaries (wall), while the k- omega model predicts well close to the boundary

**Species model**

Ansys Fluent can simulate chemical species mixing and transport by resolving conservation equations that define the convection, diffusion, and reaction sources for each component species. Many simultaneous chemical processes that occur in the fluid phase (volumetric reactions), on wall or particle surfaces, and/or in the porous zone can be explained. This section discusses species transport modelling capabilities, both with and without responses.

Fluent can model species transport and reaction using several independent models.

- Species Transport and Finite-Rate Chemistry Approach
- Non-Premixed Combustion Approach
- Premixed Combustion Approach
- Partially Premixed Combustion Approach
- Composition PDF Transport Approach

**Species Model - Non-premixed combustion**

A non-premixed model with steady diffusion flamelet and a non-adiabatic energy treatment is used. Non-premixed combustion ensures that the fuel and oxidizer enter the reaction zone as separate streams. Steady diffusion flamelet model is based on the assumption that the flame in a turbulent flow can at any time be regarded as an ensemble of small laminar diffusion flames, generally referred to as flamelets. The main advantage of this model is that the flamelets, which describe the local structure of the turbulent flame, are coupled to the turbulent flow by only a few parameters.

Both inlet diffusion and compressibility effect are taken in account for the PDF (Probability Density Function). The PDF represents a general statistical description of the turbulent reacting flow. This PDF can be considered to be proportional to the fraction of the fluid depends at each chemical species, temperature and pressure state. The shape of the assumed PDF is described either by the double delta function or the β function. The shape produced by this function depends only on the mean mixture fraction and its variance.

Operating pressure was taken as 50000Pa, where fuel and oxidizer were provided with temperature of 300K each, the maximum flamelet temperature was observed to be 2214.36K.

**Discrete phase model**

The Discrete Phase Model (DPM) is a modelling technique used in CFD simulations to follow the motion of discrete particles in a fluid flow, such as droplets, bubbles, or solid particles

Instead of defining the fluid flow, the dynamic particle model (DPM) uses a Lagrangian method. This means it monitors the motion of individual particles as they move through the fluid.

DPM is commonly used as the model of choice in simulations of multiphase flows, particularly those in which the behaviour of particles or droplets is relevant, such as spray combustion, fluidized bed reactors, or particle-laden flows. By tracking the motion of individual particles, DPM can provide exact information about the distribution, velocity, and trajectory of particles in a flow. This info. may be used to improve the design and operation of industrial processes.

Discrete Phase Model - **Injection**

This model simulates a discrete second phase in a Lagrangian frame of reference. This second phase consists of spherical particles, representing droplets dispersed in the continuous phase. Injection of fuel is done in the form of droplet with pressure swirl injection with 100 streams and particle type as droplet, the physical model is temperature dependent latent heat.

The parameters assigned during injection of fuel to the reaction zones are mentioned in the table below:

| | | |
|---|---|---|
| Material (Fuel) | | Jet A1 fuel |
| Evaporation species | | c12h23 |
| Position | Millimetre | X=37mm, Y=Z=0 |
| Axis | | X=1, Y=Z=0 |
| Temperature | Kelvin | 300K |
| Mass flow rate | Kg/sec | 0.0003Kg/s |
| Injector inner diameter | Millimetre | 1mm |
| Spray half angle | Degree | 30 |
| Upstream pressure | Pascal | 1000000Pa |
| Sheet constant | | 12 |
| Atomizer dispersion angle | Degree | 6 |

*Table 3: Fuel Injection parameters*

**Jet A1 Fuel**

The majority of turbine-powered aircraft may use the kerosine grade fuel known as Jet A-1. It has a maximum freeze point of -47 degrees C and a minimum flash point of 38 degrees C (100°F).

| | |
|---|---|
| Density | 801.8Kg/sec |
| Cp | 2066.75 J/Kg K |
| Viscosity | 0.0012676Kg/m s |
| Latent heat | 252570.5J/Kg |
| Vaporization temperature | 311K |
| Boiling point | 477.95K |
| Volatile component fraction | 100% |
| Binary diffusivity | 5.767e^-06 m^2/S |
| Diffusivity reference pressure | 101325 |

| | |
|---|---|
| Saturation Vapour pressure | 185.311Pa |
| Droplet surface tension | 0.0225N/m |

*Table 4: Fuel properties*

**BOUNDARY CONDITIONS**

This is a condition that is required to be satisfied at all or part of the boundary of a region in which the set of differential conditions is to be solved. The boundary conditions specified for the systems under study are given in the table mentioned below. The solver details used for analysis using Ansys fluent are mentioned below

ANSYS Fluent 19.2 is used for the analysis. Studies are carried out considering the flow to be steady, with velocity formulation as absolute

Pressure based segregated solver - Standard is used for solving the governing equations. Solution is second order for momentum, turbulent dissipation rate, energy, turbulent kinetic energy, mean mixture fraction, mixture fraction variance

Realizable k-ε turbulence with Standard wall condition is used for modelling viscous nature. Combustion has been modelled using non-premixed Steady diffusion Flamelet model

| S. No | SOLUTION SETUP | INPUTS |
|---|---|---|
| 1 | General | 3D, Pressure based solver, Steady state |
| 2 | Model | Energy<br>Viscous – Realizable standard wall function |
| 3 | Materials | Air - Ideal gas<br>Fuel – Jet A1 |
| 4 | Boundary conditions | Pressure inlet – 450000Pa<br>Pressure outlet – 51325 Pa<br>Operating pressure – 50000Pa |
| 5 | Solution methods | Gradient – Least square cell based<br>Turbulent kinetic energy, turbulent dissipation rate, Momentum, Energy – Second order |
| 6 | Condition for convergence | Targeted mass flow rate |

*Table 5: Boundary condition*

Based on above condition and domain simulations were carried out in Fluent and observed results are discussed

**CHAPTER 4**
**RESULT AND DISCUSSIONS**

The analysis results obtained for torch ignitor model using solver input condition as discussed in previous chapter are shown as discussed here

**NON-REACTING FLOW FIELD**

The non reacting flow simulations was carried out by the given boundary condition specifying inlet pressure, outlet pressure and operating pressure condition. Mass flow rate by using the above-mentioned boundary conditions was 0.0034kg/sec.

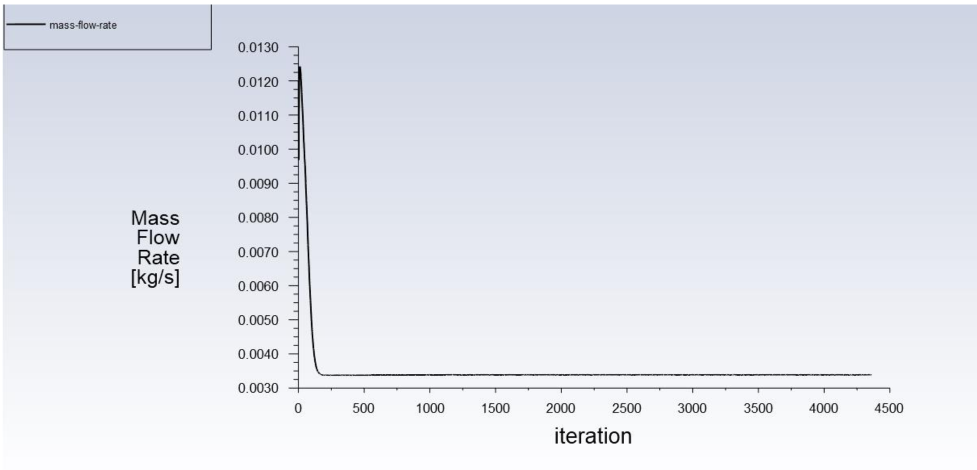



*Figure 32: Mass flow rate*

The obtained contours are shown below

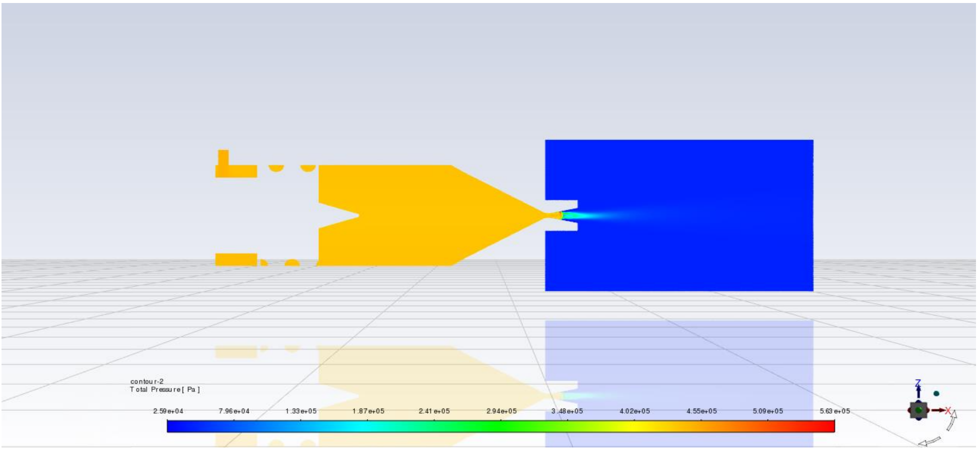



*Figure 33: Contour of Total pressure*

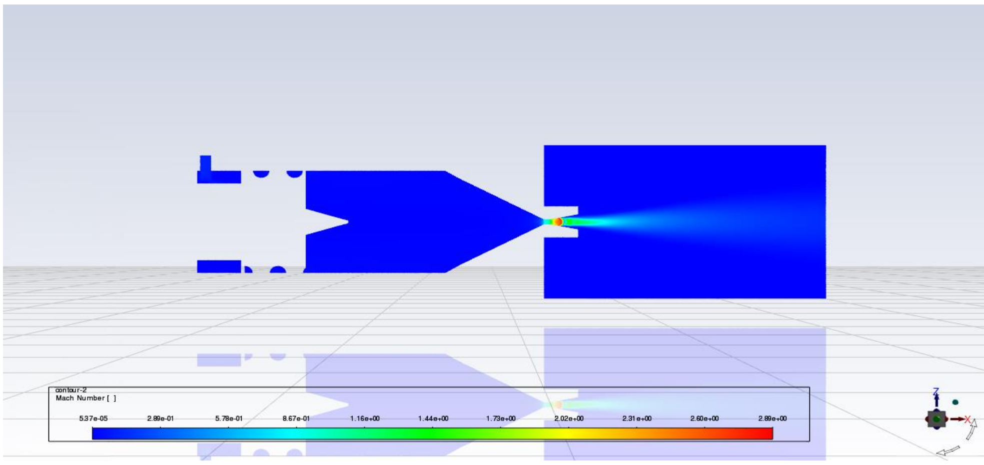



*Figure 34: Contour of Mach number*

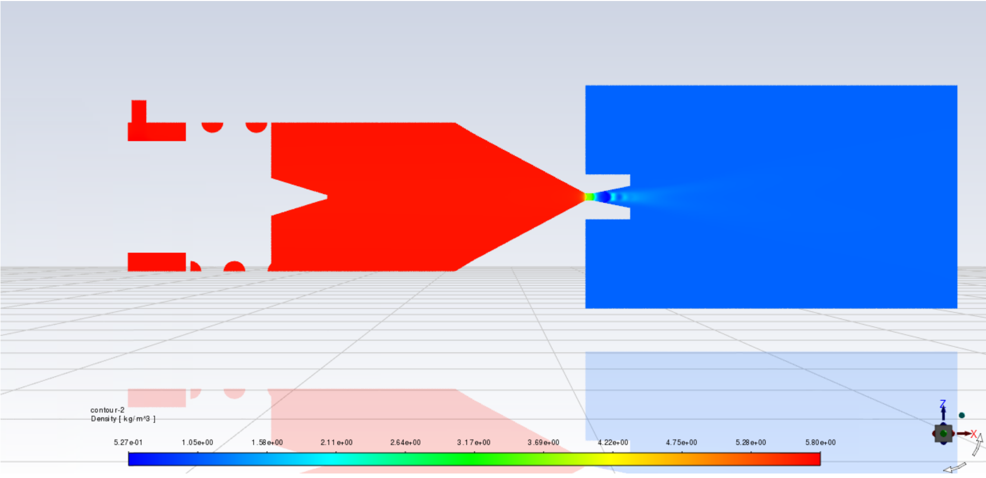



*Figure 35: Contour of Density*

fig above portrays the distribution of total pressure, Mach umber and density within the torch ignitor.
The condition acts as the preliminary process to reacting flow in order to obtain quick and better results for the further cases by mesh-to-mesh interpolation technique.

**REACTING FLOW FIELD**

Non premixed combustion model with steady state diffusion flamelet is used to create the flamelet model. Fuel is injected through pressure swirl atomizers using Discrete phase model. Below are the Total pressure, Total temperature, Density and various mass fraction contours obtained for the design case.

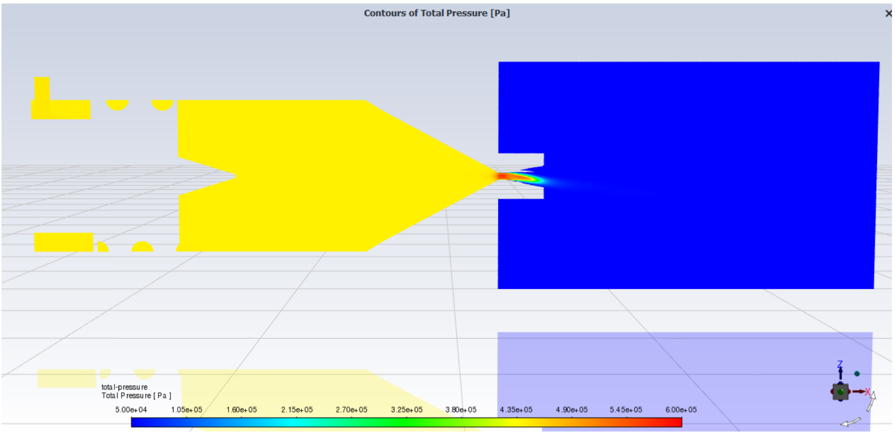



*Figure 36: Contour of Total pressure*

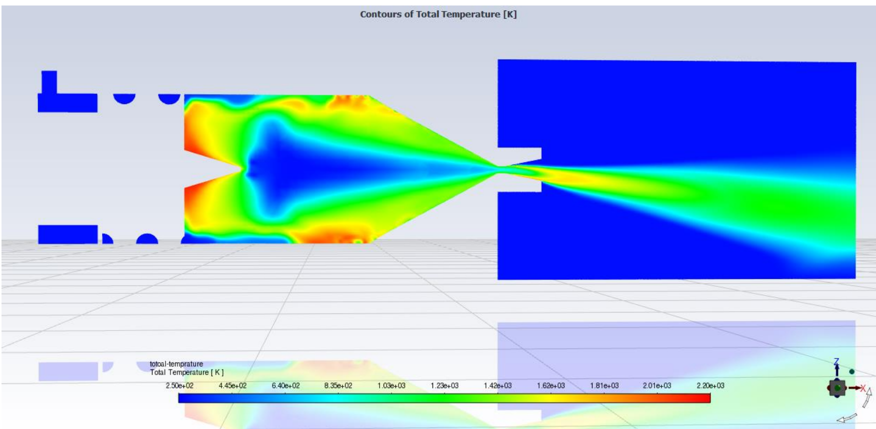



*Figure 37: Contour of Total Temperature*

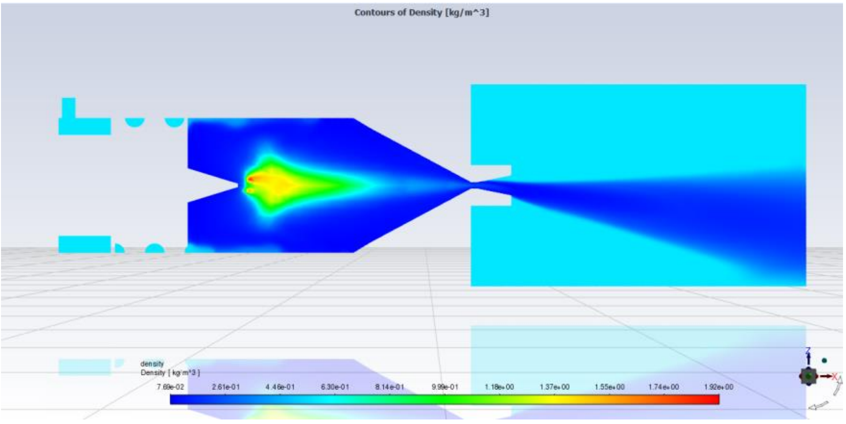



*Figure 38: Contour of Density*

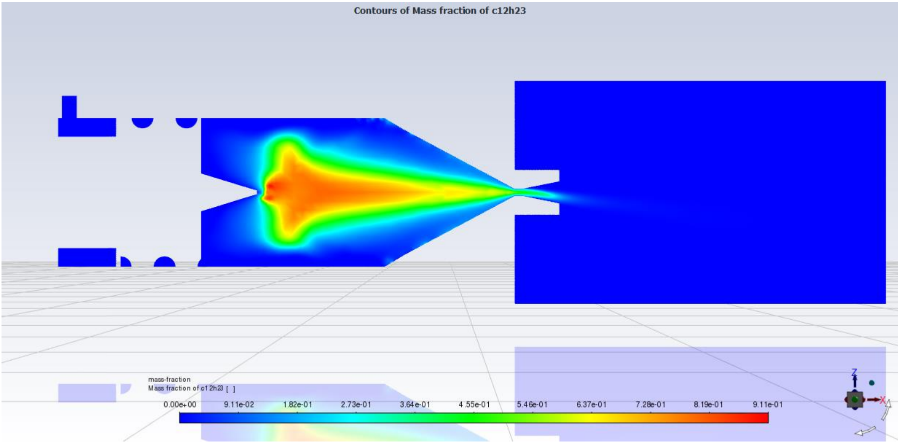



*Figure 39: Contour of Mass fraction of $c_{12}h_{23}$*

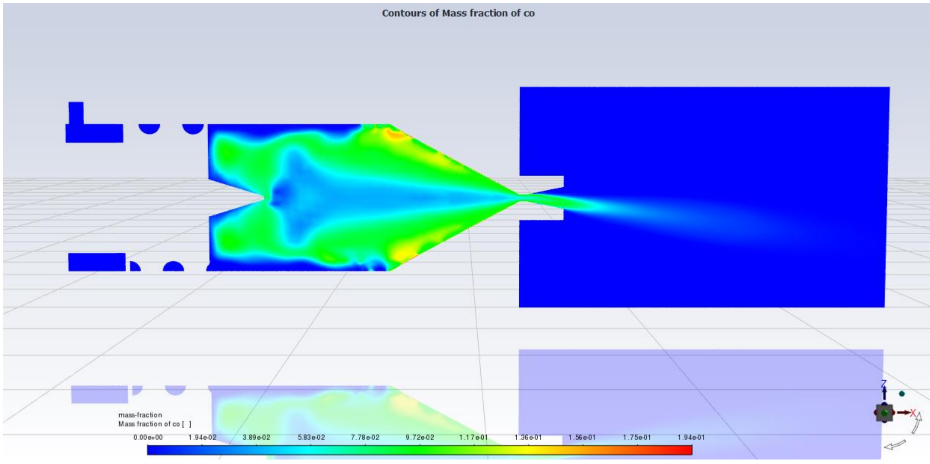



*Figure 40: Contour of mass fraction of CO*

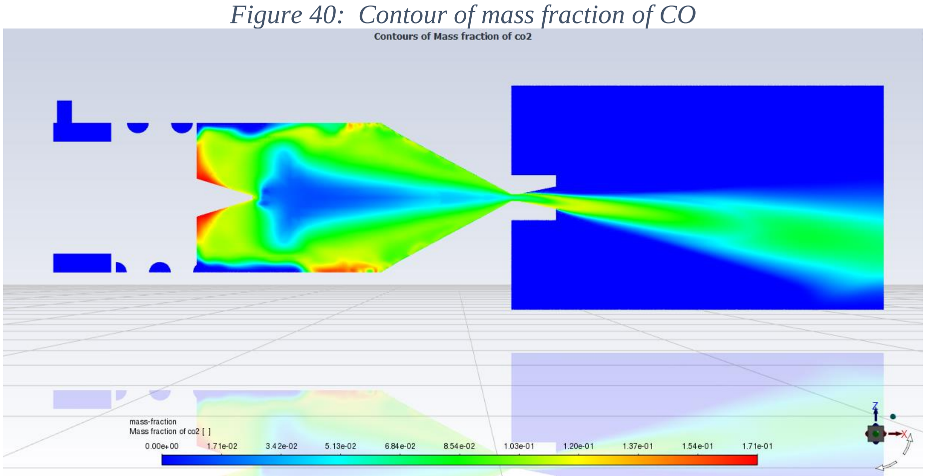



*Figure 41: Contour of mass fraction of $CO_2$*

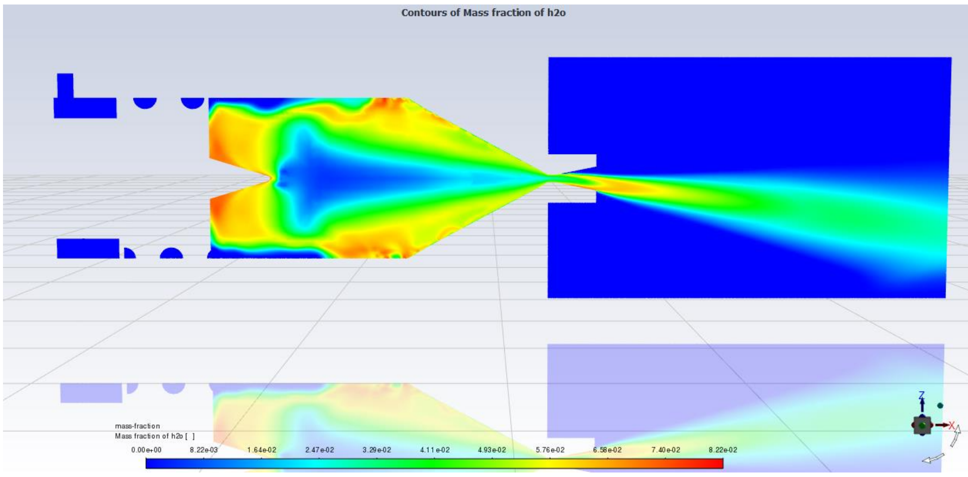



*Figure 42: Contour of mass fraction of $h_2 0$*

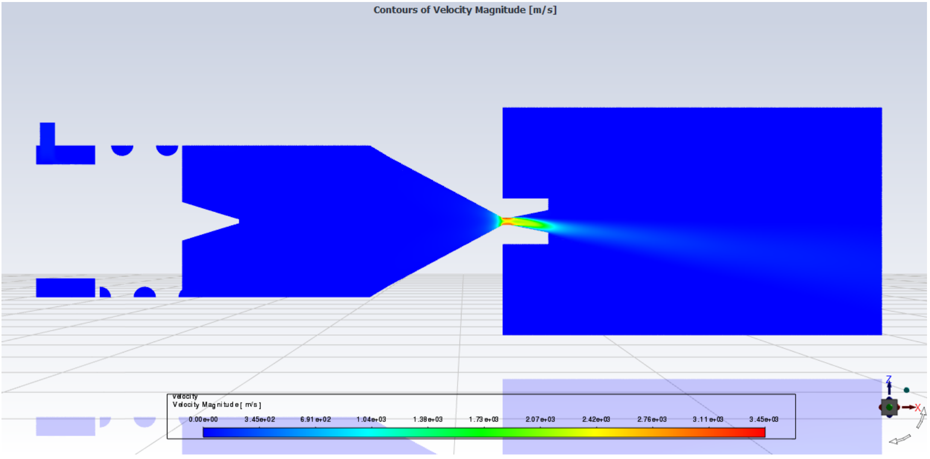



*Figure 43: Contours of velocity magnitude*

From the contour plots of mass fraction of fuel, $H_2O$, $CO_2$ and CO, it is clearly seen that the combustion products which are formed due to the reaction of fuel-air mixture are getting trapped near the fuel injecting region. From the value of mass fraction of $CO_{2,}$ it indicates that the propellant is not getting completely consumed and some unburned propellants remains, which results in higher mass fraction of CO as shown in the contour.

Also, from the contour plots of Total pressure it is observed that constant pressure is maintained inside the torch ignitor chamber. From other contours it is observed that the flow develops to the atmosphere with higher flame length.

The velocity remains constant till the end of convergent part and reaches to higher at the throat region and then it reduces as it passes through the divergent portion.

The plot showing total pressure, total temperature and velocity magnitude wrt position are shown below, the values of these parameters are taken from the convergent portion till the divergent portion of the geometry within the central part of the torch ignitor

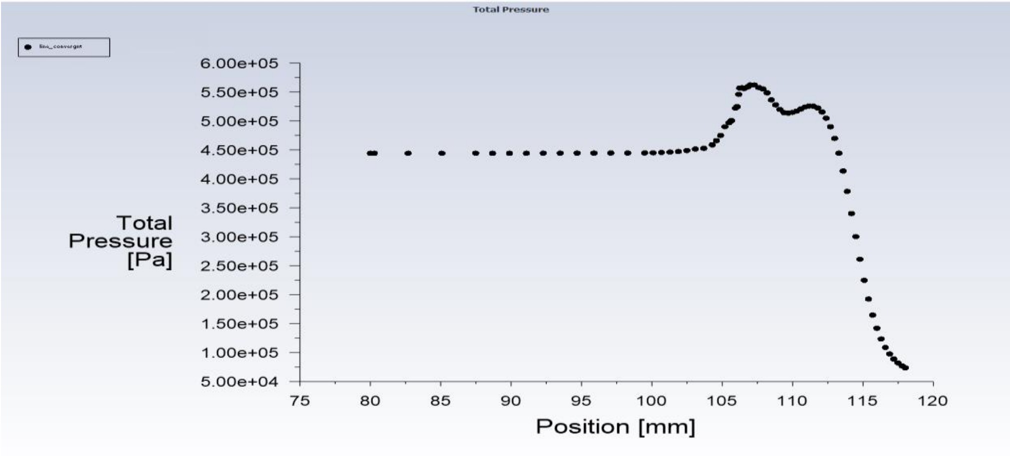



*Figure 44: Total pressure wrt position*

The pressure remains constant till the end of convergent part and then it increases within the throat and then reduces as it passes through the divergent portion

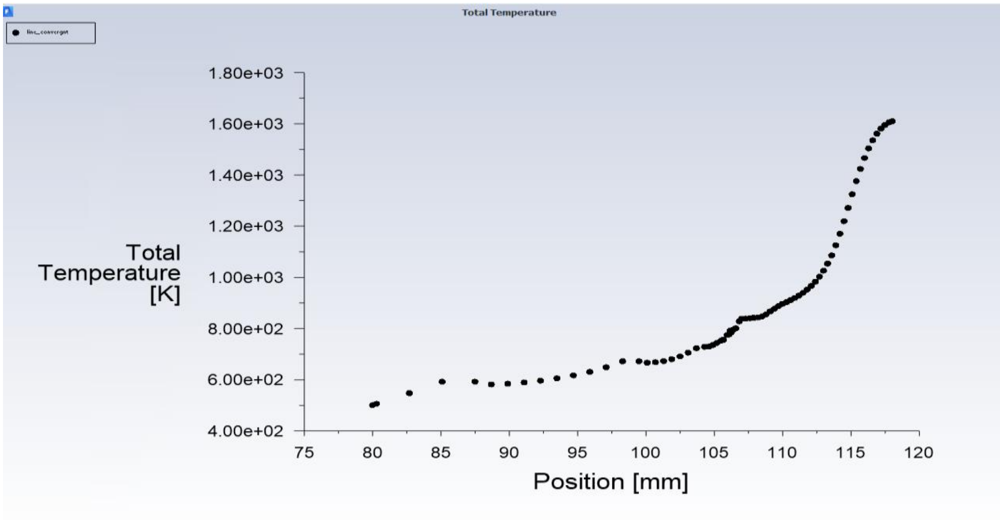



*Figure 45: Total temperature wrt Position*

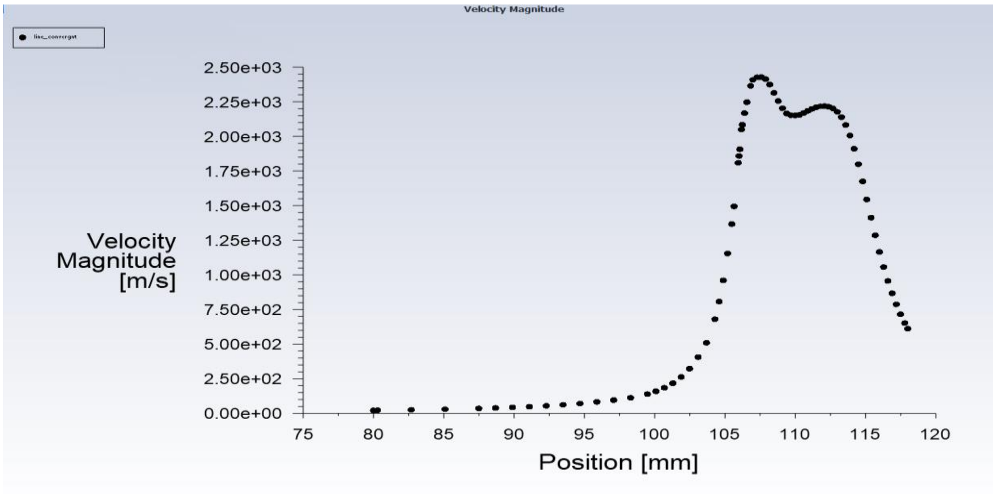



*Figure 46: Velocity wrt Position*

**CHAPTER 5**
**CONCLUSION AND FUTURE WORK**
**CONCLUSION**

Validation study with no experimental data of torch ignitor was performed using numerical simulation model in ANSYS. Used realizable k-epsilon with standard wall function turbulence model.

Design and meshing were the major challenge in this project for modelling the torch ignitor. Meshing was done using Ansys Fluent meshing with cell count between 2 million to 4 million hexacore cells

Non reacting and reacting flow CFD analysis in Ansys fluent has been carried out for the torch ignitor model with pressure swirl atomizer fuel injection for the given design condition.

The essential features of the combustor have been modelled, i.e., the viscous has been modelled using Realizable k – epsilon model with standard wall function and combustion is modelled using non premixed steady state diffusion model

Various techniques for accurate fuel – air mixture was studied, on which swirling mechanism was opted as it helps for rapid air fuel mixture and also it increases the flame propagation speed.

The concept to generate swirling motion inside the compact sized chamber was the next task to achieve, with various literature reviews it was decided to select the geometrical feature which has embedded slot within its outer structure which can import an angular rotation to the incoming air and mix with the fuel which reaches the end of the same geometrical body in axial direction.

Grid independent study of the above model was performed on different grids M1, M2, M3 and found out M2 grid having less cell count and negligible sensitivity.

Cold flow analysis (non-reacting flow) was done using mentioned boundary conditions to study the pressure distribution within the chamber, Mach number throughout the geometry along with the mass flow rate developed at the air inlet.

Reacting flow analysis was done with non-premixed combustion and steady diffusion flamelet with non-adiabatic energy treatment. A maximum flamelet temperature of 2214.36 K was achieved. Discrete phase model for injection with pressure swirl atomizer was used with droplet as particle type and time dependent latent heat as physical model.

Combustion was achieved inside the torch ignitor chamber with a maximum temperature of 2200K, which was observed near the walls of torch ignitor. It could achieve higher flame length which was an essential feature to be obtained.

**FUTURE WORK**

Testing of the model is to be done under atmospheric condition

Check the reliability of the manufactured system

Implementing the system for a real time mission

**REFERECES:**


1. Stephen N Schmotolocha, Donald H Morris, Calvin Q Morrison Jr and Robert J Pederson (Thousand Oaks, CA (US) (2004)),” Torch Igniter”, US Patent No:US006912857B2, June 2005
2. Teru Morishita (Shizuoka, Japan), “Ignition Torch”, US Patent No:4215979, August 1980.
3. Noel Parker Coupe, North Ferriby, England, assignor to Blackburn and General Aircraft Limited, Brough, “igniters for gas turbine engines, combustion heaters, thermal de-icing plants, Dec. 23, 1958
4. Yanlin chen, Xiangrong Li, Shuainan Shi, Qingxu Zhao,” Effects of intake swirl on the fuel/air mixing and combustion performance in a lateral swirl combustion system for direct injection diesel engines”, 15 Feb 2021.
5. Atticus J Vadera, Liquid Rocket Engine Torch Igniter Feed System
6. Paul Prochnicki, Jan Fessl, Michael J. Moruzzi, Eric Perry, and John A. Targonski. Hydra: Development of a Liquid Rocket Engine Test Stand and Feed System.
7. Kesiany Maxima de Souza, Olexiy Shynkarenko, Development of a measurement system of temperature and pressure in the combustion chamber of a torch ignition system.
8. . Olexiy Shynkarenko, Domenico Simone, Jungpyo Lee and Artur E. M. Bertoldi, Experimental and Numerical Study of the Flammability Limits in a CH4/O2 Torch Ignition System
9. . Olexiy Shynkarenko and Domenico Simone, Oxygen–Methane Torch Ignition System for Aerospace Applications.
10. . Kenichi Takita Tomokazu Uemoto Takahiro Sato Yiguang Ju Goro Masuyaand Katsura Ohwaki, Ignition Characteristics of Plasma Torch for Hydrogen Jet in an Airstream.
11. **.** Luis Eduardo Sanchez, Development and Testing of Oxygen/Methane Torch Igniter Technologies for Propulsion Systems.
12. Flores, Jesus Roberto, Development and testing of an ignition physics test facility and an oxygen/methane swirl torch igniter
13. John D Anderson, "Computational fluid dynamics: The basics with applications", McGraw Hill, 1995
14. Arthur. H. Lefebvre., "Gas Turbine Combustion", 2nd Edition, 1999.
15. Arthur H. Lefebvre and Dilip R. Ballal, GasTurbine Combustion Alternative Fuels And Emissions", Third Edition.
16. Meherwan P Boyce, "Gas Turbine Engineering Handbook, third edition 28 April, 2006