International Journal of Scientific Engineering and Research (IJSER)
ISSN (Online): 2347-3878
Index Copernicus Value (2015): 62.86 | Impact Factor (2015): 3.791

# Design and Optimization of Combustion Air Engine Simulator

**Archith V[1], Arqam Abdullah[2], Ashik R[3], Adith Sagar Narayan[4]**

[1]Department of Mechanical Engineering, Nitte Meenakshi Institute of Technology, PB No-6429, Yelahanka, Bangalore-560064
[2]Department of Mechanical Engineering, Nitte Meenakshi Institute of Technology, PB No-6429, Yelahanka, Bangalore-560064
[3]Department of Mechanical Engineering, Nitte Meenakshi Institute of Technology, PB No-6429, Yelahanka, Bangalore-560064
[4]Department of Mechanical Engineering, Nitte Meenakshi Institute of Technology, PB No-6429, Yelahanka, Bangalore-560064

**Abstract:** ***Combustion air engine simulators have experienced a growing interest in displaying the working model of gas turbine engine. The simulators are used in various applications such as debris monitoring, and visual analysis of the working of the gas turbines. The design and simulation consists of simplifying the complex model into an exact miniaturized working model. This paper is focused on the study of behavior of gases and parameters affecting it in the combustion chamber of an air engine simulator. The various parameters include design of the combustion chamber for obtaining uniform generation of the swirl, obtaining uniform mixture of air and fuel for complete combustion and maintenance of pressure gradient for the generation of high thrust at the outlet. The design and optimization is carried out through ANSYS fluent software.***

**Keywords:** Simulator, miniaturized, swirl, ANSYS fluent, gradient

## Introduction

The design of the combustion chamber includes miniaturization of the existing model and simplification in every possible way, such that the result is not affected, and the exact behavior of the model is obtained. Starting from the most complex structure the turbine, as it is difficult to manufacture turbine of smaller size and issues related to its mounting on the axis, it was decided to remove the turbine but restructure the model in order to obtain the results similar to that of the presence of turbine, if the turbine is absent even the compressor is absent. The immediate solution to this problem is an air blower, the air blower compresses the air and discharges compressed air at its outlet and for swirling of the air due to the presence of turbine, we can adjust the position of the blower at an angle such that the air in the chamber swirls in a manner similar to that of an air combustion engine chamber.

High octane fuels are used in air combustion engine which later gets atomized in the chamber, but in our model instead of atomization we inject direct gaseous fuel, fuel such as LPG as it is easily available and comparatively a high calorific value. The air-fuel ratio of air combustion engine is 60:1 but LPG requires 30:1.The structure of the chamber should be such that it should favor a pressure gradient along the length of the chamber such that the gases flow out of the chamber.

## Literature Survey

The combustion chamber of gas turbine unit is one of the most critical components to be designed. The reason behind the designing of the gas turbine combustion chamber being critically important is a need for stable operation. The reason behind the designing of the gas turbine combustion chamber being critically important is a need for stable operation over wide range of air/fuel ratios [1]. In practice, to ensure complete combustion, excess air is supplied beyond that theoretically required for full oxidization of the fuel. This express as a percentage of the theoretical air needed i.e. 10% excess air is 1.1 times the theoretical air quantity [2]. LPG is typically a mixture of several gases in varying proportions. Major constituent gases are propane ($C_3H_8$) and butane ($C_4H_{10}$), with minor quantities of propane ($C_3H_6$), various butanes ($C_4H_8$), iso-butane, and small amounts of ethane ($C_2H_6$) [3]. Swirl level had a significant influence on the combustion process and exhaust emissions. When swirl ratio was increased up to a certain level, a more intense pre-mixed combustion phase was observed, and the diffusion controlled combustion phase was also improved. The swirl vanes are design in such a manner that flow turns 30 degree from the previous path [4]. The design of annular type of combustion chamber for small gas turbine application. The CFD simulation is carried out for the designed chamber using commercial CFD tool; ANSYS CFX [1]. The combustion of propane with air was analyzed in a burner element and the effect of the equivalence ratio and oxygen percentage in air investigated, for different numerical values. Combustion was simulated for the fuel mass flow rate resulting in the same heat transfer rate to the combustion chamber in each case [5].The reason for the lower combustion duration is the higher laminar burning velocity of LPG (0.46 m/s) when compared with gasoline (0.42m/s) and lowered the emission also. On an energy basis LPG has a lower carbon content than gasoline or diesel fuel [6]. The different modes of Combustion are premixed combustion, diffusion combustion and mixed mode combustion. In premixed combustion air and fuel are premixed to the required stoichiometry before burning [7].

## Theoretical Calculations

### 3.1 Stoichiometric Calculations

The combustion calculations are done without considering nitrox formation. The speed of the blower is 3.3 $m^3$ /min. atmospheric air consists of only 20% of oxygen. LPG in India consists of 60% butane and 40% propane by volume. Their respective reaction with oxygen is as follows:

$C_3H_8 + 5\,O_2 \longrightarrow 3CO_2 + 4\,H_2O$ + energy
1Vol. 5Vol.
$C_4H_{10} + 13/2\,O_2 \longrightarrow 4CO_2 + 5H_2O$ + energy
1Vol. 6.5Vol.

#### 3.1.1 For butane

Every 1 unit volume of butane requires 6.5 unit volumes of oxygen.
Every 0.6 unit volume of butane requires 3.9 unit volumes of oxygen.

#### 3.1.2 For propane

Every 1 unit volume of propane requires 5 unit volumes of oxygen.
Every 0.4 unit volume of propane requires 2 unit volumes of oxygen.
Therefore total volumes of oxygen required is 3.9 + 2 = 5.9 volumes.

As atmospheric air consists of only 20% of oxygen by volume, the amount of air required for this will be 29.5 volumes of air.

Therefore the theoretical air-fuel ratio is 29.5:1.

For complete combustion excess air is to be provided so the actual air fuel ratio is determined to be 31:1.

### 3.2 Adiabatic Flame Temperature

$C_3H_8 + 5\,O_2 \longrightarrow 3CO_2 + 4H_2O$ + energy
1moles 5moles 3moles 4moles
0.4moles 2moles 1.2 moles 1.6moles
Enthalpy of formation of propane $h_f C_3H_8$
$h_f C_3H_8 = h_f CO_2 + h_f H_2O - h_f C_3H_8$

**Table 1:** Enthalpy of formation

| *Enthalpy of formation $h_f$* | *KJ/K mole* |
|---|---|
| $CO_2$ | -393.522 |
| $H_2O$ | -241.827 |
| $C_3H_8$ | -103.85 |
| $C_4H_{10}$ | -125.29 |

#### 3.2.1 Enthalpy of formation of Propane

$h_f C_3H_8 = h_f CO_2 + h_f H_2O - h_f C_3H_8$
$= 1.2\,(-393.522) + 1.6\,(-241.827) - 0.4(-103.85)$
$= -817.61$ kJ

#### 3.2.2 Enthalpy of formation of Butane

$C_4H_{10} + 13/2\,O_2 \longrightarrow 4CO_2 + 5\,H_2O$
1moles 6.5 moles 4moles 5moles
0.6moles 3.9moles 2.4moles 3moles
$h_f C_4H_{10} = h_f CO_2 + h_f H_2O - h_f C_4H_{10}$
$= 2.4(-393.522) + 3(-241.287) - 0.6(-125.29)$
$= -1594.76$KJ

The exit state for flame temperature is specified by:
$\sum n_c h_c = h_f C_3H_8 + h_f C_4H_{10}$
$= -817.61 + (-1594.76)$ KJ
$= (-2412.36)$ KJ

**Table 2:** Obtaining the solutions using averaging specific heat values at 500K

| *GAS* | *Specific heat $C_p$ KJ/mole* |
|---|---|
| $CO_2$ | 0.045 |
| $H_2O$ | 0.035 |
| $O_2$ | 0.030 |
| $N_2$ | 0.030 |

$\sum n_c h_c = \Delta T\{C_p(CO_2) + C_p(H_2O) + C_p O_2 + C_p(N_2)$

$= \Delta T \{(0.045)\, 3.6 + (0.035) 4.6 + (0.030) 5.9$

$+ (0.030)\, 18.8\}$

$2412.36 = 1.064\Delta T$

Initial temperature 298K

Now, $\Delta T = T_{final} - T_{initial}$

$T_{final} = T_{adiabatic} = 2565.26K$

## Modeling

The Geometry of the combustion chamber was created using ANSYS WORKBENCH 15.0

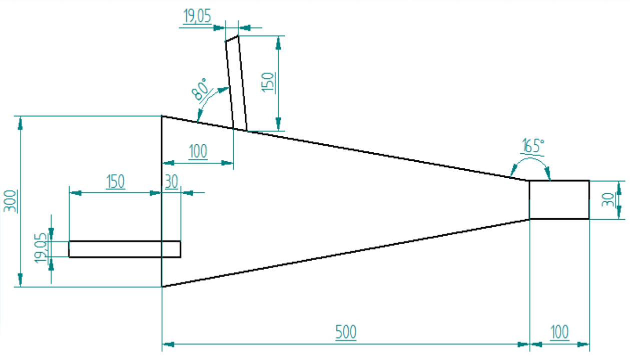



**Figure 1:** Model with dimension

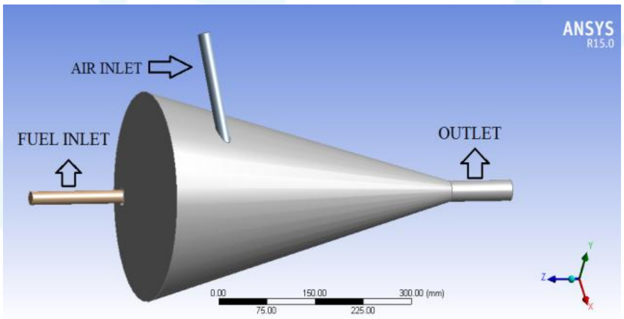



**Figure 2:** Model of the chamber

**Table 3:** Design parameters

| | |
|---|---|
| Length of the chamber | 500 mm |
| Inlet diameter of chamber | 300 mm |
| Outlet diameter of chamber | 30 mm |
| Length of the outlet | 100 mm |
| Air inlet diameter | 19.05 mm (3/4 inch) |
| Outlet diameter | 30 mm |
| Fuel inlet diameter | 19.05 mm (3/4 inch) |

## Meshing

ANSYS ICEM CFD meshing is used. After the modeling is completed the meshing is to be done. The module used to perform meshing is fluid flow (fluent). The meshing method used here is Automatic method and the mesh type for all components is hexahedral and tetrahedron type.

**Table 4:** Types of meshing

| *Component* | *Type of mesh* |
|---|---|
| Inlet-air | Hexahedral mesh |

| | |
|---|---|
| Inlet-fuel | Hexahedral mesh |
| Combustion chamber | Tetrahedron mesh |
| Outlet | Hexahedral mesh |

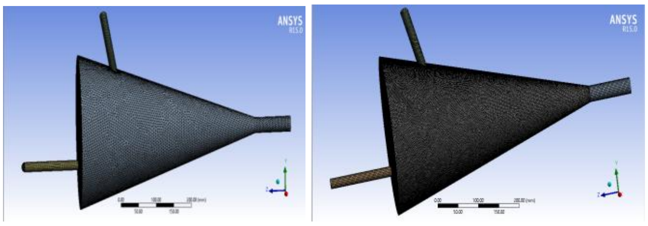


**Figure 3:** Mesh size of 6mm and mesh size of 3mm

**Table 5:** Characteristics of Mesh

| *Mesh body sizing (in mm)* | *Mesh quality (orthogonal) (in mm)* | *Mesh quality (skewness) (in mm)* | *No. Of elements* | *No. of nodes* |
|---|---|---|---|---|
| 6 | 0.86 | 0.221 | 568290 | 108176 |
| 3 | 0.86 | 0.217 | 4302413 | 757988 |

Skewness values of the cells-it ranges from 0 to 1, close to 1 is low quality.
Orthogonal quality of the cells-it ranges from 0 to 1, close to 1 is high quality.

## Boundary Conditions

**6.1 Air-Velocity Inlet**
**6.2 Fuel-Velocity Inlet**
**6.3 Outlet-walls**

Specification of the boundary zones has to be done in WORKBENCH only, as there is no possibility to specify the boundary zones in FLUENT. Therefore proper care has to be taken while defining the boundary conditions in WORKBENCH. With all the zones defined properly the mesh is exported to the solver. The solver used in this problem is ANSYS FLUENT. The exported mesh file is read in Fluent for solving the problem.

**Table 6:** Various velocities of air and fuel inlet for 19.05 mm (3/4 inch) pipe diameter

| *Discharge rate of blower cubic meter/min* | *Discharge rate of fuel cubic meter/min* | *Air-inlet m/s* | *fuel-inlet m/s* |
|---|---|---|---|
| 3.3 | 0.106451613 | 192.9824561 | 6.225240521 |
| 3.4 | 0.109677419 | 198.8304094 | 6.413884173 |
| 3.5 | 0.112903226 | 204.6783626 | 6.602527825 |
| 3.6 | 0.116129032 | 210.5263158 | 6.791171477 |
| 3.7 | 0.119354839 | 216.374269 | 6.979815129 |
| 3.8 | 0.122580645 | 222.2222222 | 7.168458781 |
| 3.9 | 0.125806452 | 228.0701754 | 7.357102434 |
| 4 | 0.129032258 | 233.9181287 | 7.545746086 |
| 5.1 | 0.164516129 | 298.245614 | 9.620826259 |

## Solving

K-epsilon (k-ε) turbulence model is the most common model used in Computational Fluid Dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. It is a two equation model which gives a general description of turbulence by means of two transport equations (PDEs). The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows.

## Results and Discussion

### 8.1 Air inlet optimization

For obtaining a swirl the air inlet must be aligned tangential to the circumference of the chamber and the particles of the air experience irrotational motion and hence velocity is inversely proportional to its radius, so only for a limited range of alignment swirl is obtained. The alignment trials are as follows:

**Table 7:** Different positions of air inlet

| *Sl.No.* | *Rotation about x-axis (in deg)* | *Rotation about y-axis(in deg)* | *Result of swirl* |
|---|---|---|---|
| 1 | 30 | 100 | No |
| 2 | 35 | 100 | Yes |
| 3 | 40 | 100 | Yes |
| 4 | 45 | 100 | No |

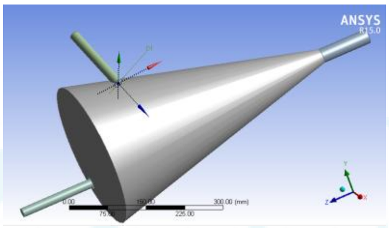
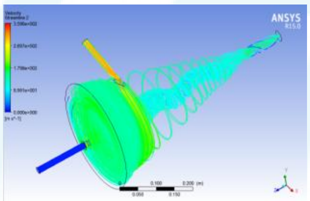
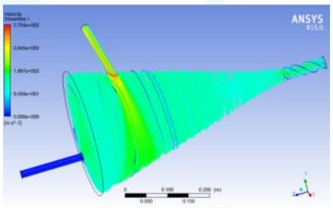


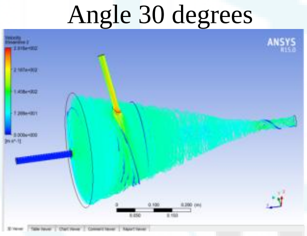

**Figure 4:** Optimum position of air inlet



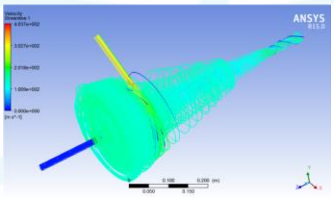
Angle 30 degrees



Angle 35 degrees



Angle 40 degrees



Angle 45 degrees

**Figure 5:** Different positions of air inlet

At an angle below 30 degree no uniform swirl was obtained. At an angle of 30 degree flow begins to establish, 35- 40 degree more uniform swirl was obtained. For angle above 40 degrees swirl flow begins to distort. Therefore the optimized angle is from 35-40 degrees. Hence the optimum angle of 35 degree was selected.

### 8.2 Fuel-inlet optimization

Fuel inlet is given along the length of the chamber, due to its low velocity the Particles of fuel flow along with the air particles. The fuel inlet is placed more towards the periphery of the chamber. It is essential for fuel particles to obtain maximum momentum from the air particles for thorough mixing of fuel with air for proper combustion. Therefore fuel inlet must be placed at a point where maximum momentum can be obtained, this is determined by positioning the fuel inlet various points and various depth into the chamber. Also the position of fuel inlet should be such that the particles of fuel do not swirl in a direction opposite to that of air. The fuel inlet is given from the face of the chamber (diameter 300 mm).

**Table 8:** Various fuel inlet position

| *Sl.No.* | *Depth of Fuel-inlet into the chamber (in mm)* | *Result of swirl* |
|---|---|---|
| 1 | 20 | Yes |
| 2 | 30 | Yes (optimized) |
| 3 | 50 | Yes |

International Journal of Scientific Engineering and Research (IJSER)
ISSN (Online): 2347-3878
Index Copernicus Value (2015): 62.86 | Impact Factor (2015): 3.791

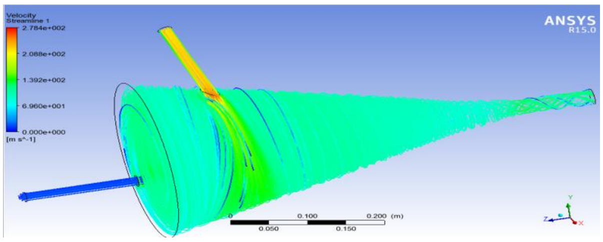
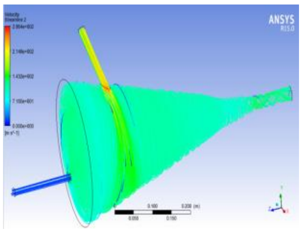
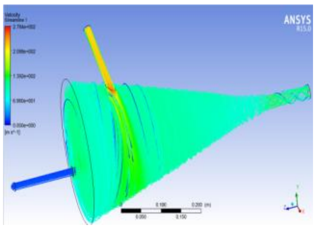



**Figure 6:** optimized fuel-inlet



Depth 20 mm



Depth 30 mm

Depth 50 mm

**Figure 7:** Fuel inlet for different depths

From the analysis it was initially concluded that the fuel inlet must be placed at a point far from the air inlet point, and hence a distance of 80 mm from x-axis and 80 mm from y-axis was determined as shown in figure. As the depth of the fuel inlet increases in the chamber more uniform mixing of air and fuel was obtained, up to the depth of 30 mm. The value of depth above 30 mm the flow begins to degenerate.

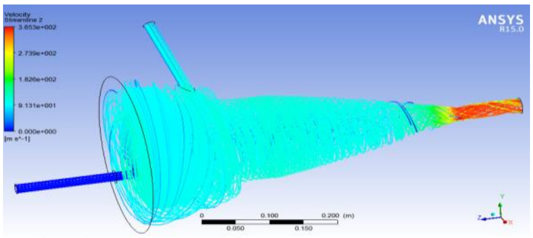
Hence the optimal depth of fuel inlet is 30 mm inside the chamber far away from the air inlet point.

### 8.3 Outlet Optimization

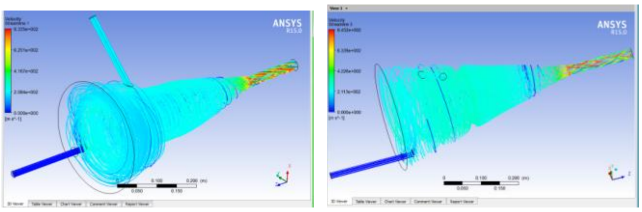
The length of the outlet cylinder plays a dominant role in establishing the swirl in the tapered chamber. Following is the analysis for determining the flow in the chamber for various length of the outlet.

**Table 9:** Different length of outlet

| *Sl.No.* | *Length of outlet (in mm)* | *Reverse flow* |
|---|---|---|
| 1 | 100 | No Reverse flow |
| 2 | 150 | Partial Reverse Flow |
| 3 | 175 | Obtained |




100 mm



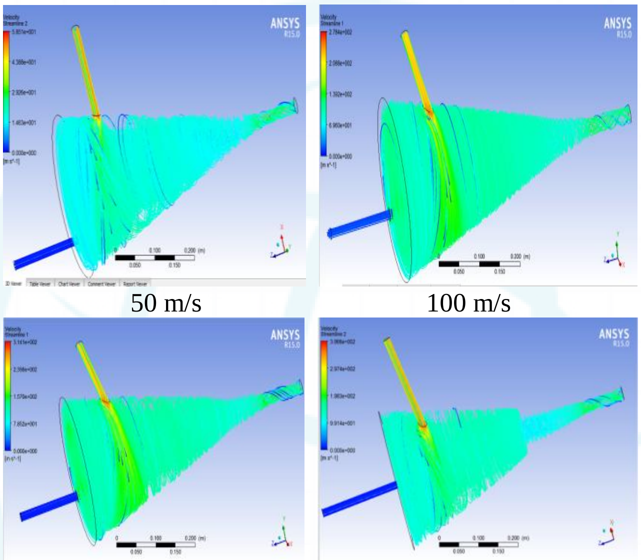

150 mm 175 mm

**Figure 8:** Different length of outlet

From the analysis it is concluded that as the length of the exhaust increases, reverse flow is experienced, therefore a value of 100 mm is selected as the length of the exhaust.

### 8.4 Swirl generation before combustion

Only for a limited range of velocities of air the swirl was obtained. The maximum velocity of air for which the swirl was obtained is 300 m/s. Following swirl was obtained for a given set of velocities of air and their corresponding velocities of fuel as required for combustion, the flow is as follows:




300 m/s 330 m/s

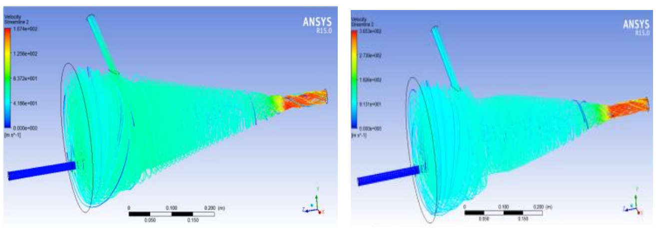
**Figure 9:** Swirls for different velocities before combustion

The minimum value of air for which the swirl is obtained is 50 m/s. The maximum value of air for which the swirl is obtained is 300 m/s. After this no proper swirl is obtained. For the generation of a high thrust at the outlet a high velocity of air is required at the inlet. Therefore a velocity of 300 m/s is selected.

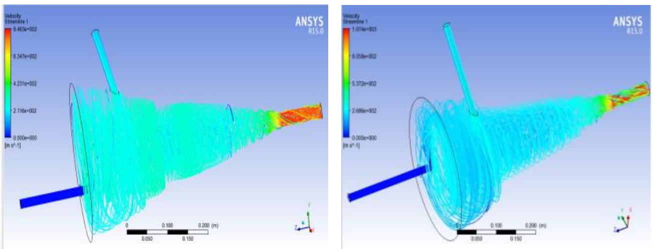
### 8.5 Swirl Generation after Combustion

Only for a limited range of velocities of air the swirl was obtained. When simulation for combustion is carried out the maximum velocity of air for which the swirl was obtained differed from that of non-combustion values.

The maximum velocity of air for which the swirl was obtained is 230 m/s. Following swirl was obtained for a given velocities of air and their corresponding velocities of fuel, the flow is as follows:




50 m/s 100 m/s



230 m/s 300 m/s

**Figure 10:** Swirls for different velocities after combustion

The minimum value of air for which swirl is obtained after combustion is 50 m/s and the maximum value is 230 m/s. After 230 m/s the swirl was towards center. Therefore for a high thrust a value of 230 m/s is finalized.

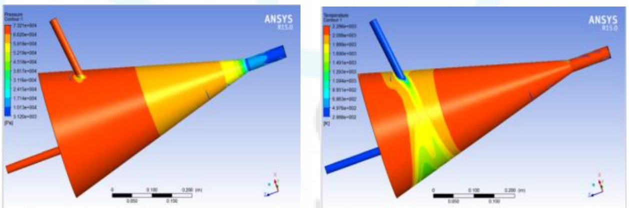
### 8.6 Pressure and Temperature values

**Table 10:** For all optimized input parameters the following result was obtained

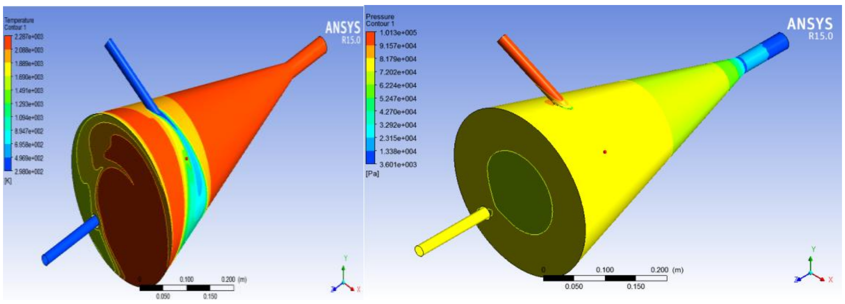
| *Mesh Size* | *Average Temperature at the Walls (in K)* | *Maximum Temperature at the Walls (in K)* | *Average Temperature at the Outlet (in K)* | *Maximum Temperature at Outlet (in K)* |
|---|---|---|---|---|
| 6mm | 1905.57 | 2287.29 | 2180.04 | 2180.04 |
| 3mm | 1892.63 | 2287.44 | 2190.17 | 2190.2 |

| *Mesh Size* | *Average Pressure at the Walls (in Pa)* | *Maximum Pressure at the Walls (in Pa)* |
|---|---|---|
| 6mm | 70234.6 | 73179.7 |
| 3mm | 73375.2 | 102906 |

According to the grid independent study the values for different mesh sizes were obtained to be similar. Temperature and pressure values were calculated near the walls inside the chamber and at the outlet correspondingly average values and maximum values were determined and all the simulated values were approximately close to the theoretical values.



Pressure Contour for 6 mm mesh Size Temperature contour for 6 mm mesh size



Pressure Contour for 3mm mesh size Temperature Contour for 3mm mesh size

**Figure 11:** Pressure and temperature contours for various mesh size

### 8.7 Convergence History

The following graphs present the convergence history of combustion chamber at different types of meshing sizes.

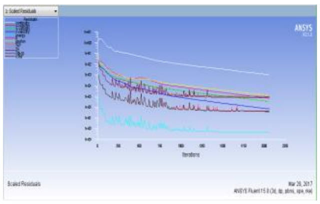


**Figure 12:** Convergence of inlet air angle 30 degree

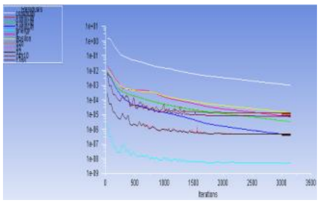


**Figure 13:** Convergence of inlet air angle 35 degree

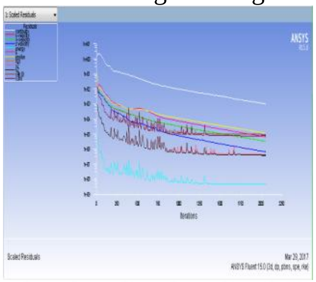


**Figure 14:** Convergence of inlet air angle 40 degree

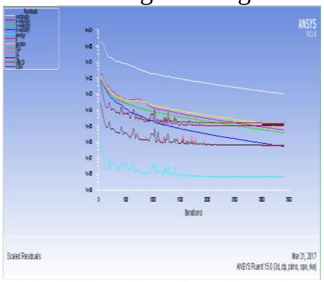


**Figure 15:** Convergence of inlet air angle 45 degree

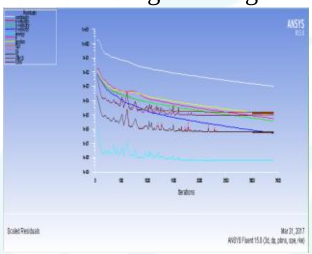


**Figure 16:** Convergence of optimized fuel inlet

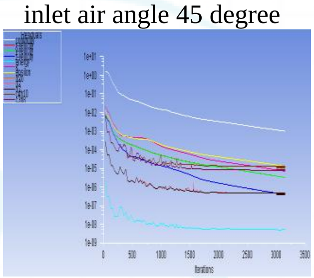


**Figure 17:** Convergence of fuel inlet depth 20 mm

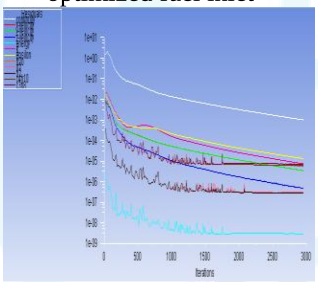


**Figure 18:** Convergence of fuel inlet depth 30 mm

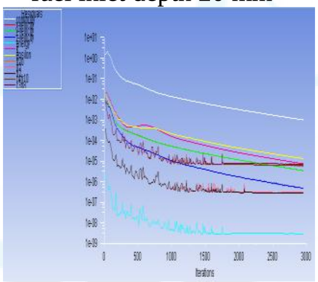


**Figure 19:** Convergence of fuel inlet depth 50 mm

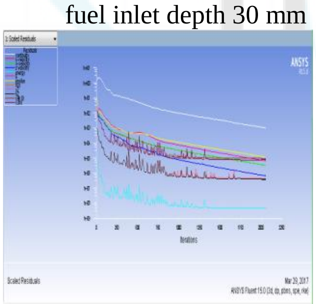


**Figure 20:** Convergence of outlet length 100 mm

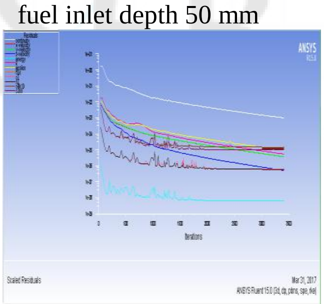


**Figure 21:** Convergence outlet length 150 mm

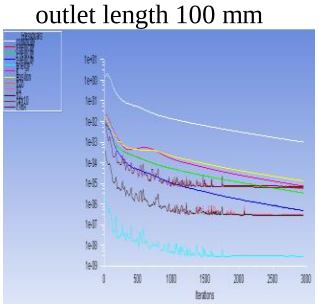


**Figure 22:** Convergence of outlet length 175 mm

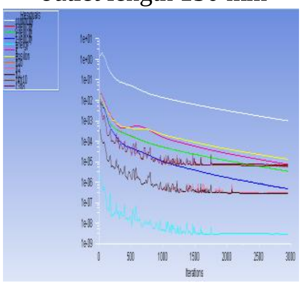


**Figure 23:** Convergence of air velocity 50 m/s before

combustion

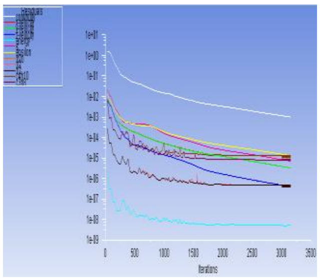


**Figure 24:** Convergence of air velocity 100 m/s before combustion

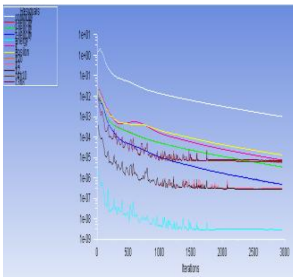


**Figure 25:** Convergence of air velocity 300 m/s before combustion

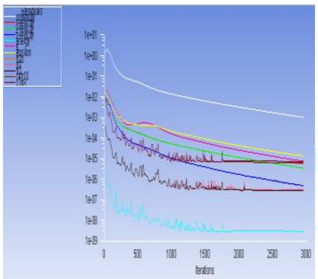


**Figure 26:** Convergence of air velocity 330 m/s before combustion

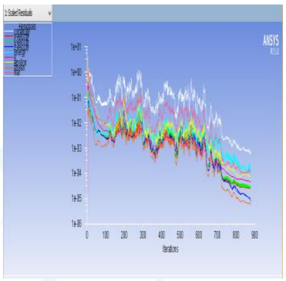


**Figure 27:** Convergence of air velocity 50 m/s after combustion

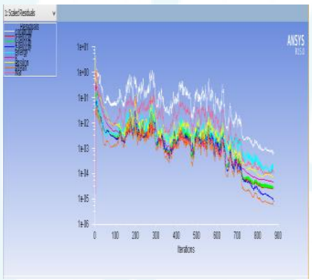


**Figure 28:** Convergence of air velocity 100 m/s after combustion

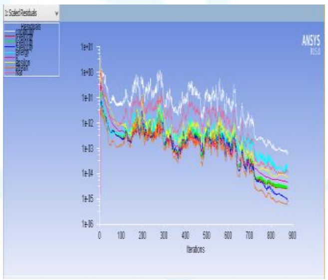


**Figure 29:** Convergence of air velocity 230 m/s after combustion

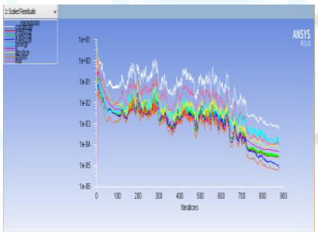


**Figure 30:** Convergence of air velocity 300 m/s after combustion

## Conclusion

After the thorough analysis of all the parameters affecting the flow of gases in the combustion chamber and considering various design parameters required for the successful working of the air engine simulator, we conclude that the shape of the chamber should be tapered for maintaining the pressure gradient. The angle of the air inlet is an important parameter in determining the generation of swirl. Therefore it is placed at an angle of 35-40 deg. The position of fuel inlet is an important criteria for thorough mixing of air and fuel, so it is placed at a point far away from the air inlet at 80 mm from x-axis and 80 mm from y-axis and at a depth 30 mm inside the chamber such that the fuel particles experience maximum momentum for uniform mixing. The length of the outlet beyond 100 mm leads to reverse flow, so in order to generate maximum thrust and to avoid revere flow the length of the outlet is taken as 100 mm.

## Future Scope

Our research is based on building a prototype rather than a full-fledged model thereby checking the feasibility of actual model. This can be used in debris monitoring applications of aircraft engine. By employing this system the life of the engine of an aircraft can be predicted with minimum expense.

## References

[1] K. V. Chaudhari, D. B. Kulshreshtha, S.A. Channiwala, "Design and CFD Simulation of Annular Combustion Chamber with Kerosene as Fuel for 20 kW Gas Turbine Engine", Vol. 2, Issue 6, pp.1641-1645, November- December 2012.
[2] [2]Muhammad Aminuddin Bin AbdRahman, "Application of fluent to predict combustion performance for pure propane", thesis submitted in fulfillment of the requirements for the Award of the degree of bachelor of chemical engineering (gas technology)
[3] AlbelaH.Pundkar, S.M. Lawankar, Dr. Sameer Deshmukh, "Performance and emissions of lpg fueled internal combustion engine", a review, International journal of scientific & engineering research volume 3, issue 3, march -2012 Issn 2229-5518
[4] Syed Alay Hashim, "Design and fabrication of an annular combustion chamber for the micro gas turbine engine applications", International journal of engineering research & technology (IJERT) vol. 2 issue 8, august – 2013 IJERT Issn: 2278-0181
[5] C.E.L. Pinho, J.M.P.Q. Delgado, R. Pilão, J. Conde, and C. Pinho, "Numerical Study of Propane-Air Mixture Combustion in a Burner Element", Vols. 273-276, pp 144-149, February (2008).
[6] OrhanDurgun, HakanBayraktar, "Investigating the effects of Lpg on spark ignition engine combustion and performance energy conversion and management" volume 46, issues 13-14, august 2005, pp 2317-2333
[7] Aruna Devadiga, Prof.Dr. T. NageswaraRao, "Optimizing Bunsen burner Performance Using CFD Analysis", Vol. 3, Issue. 5, pp-2773-2785 ISSN: 2249-6645, Sep - Oct. 2013.

## Author Profile

**Archith V** Pursuing Bachelor of Engineering in Mechanical Engineering in the year 2017 at Nitte Meenakshi Institute of Technology, Bengaluru, India. His areas of interest in Project management, Mechanical Design, Automobile

**Arqam Abdullah** Pursuing Bachelor of Engineering in Mechanical Engineering in the year 2017 at Nitte Meenakshi Institute of Technology, Bengaluru, India. His areas of interest in Mechanical Design, Production.

**Ashik R** Pursuing Bachelor of Engineering in Mechanical Engineering in the year 2017 at Nitte Meenakshi Institute of Technology, Bengaluru, India. His areas of interest in Mechanical Design, Automobile, Manufacturing.

**Adith Sagar Narayan** Pursuing Bachelor of Engineering in Mechanical Engineering in the year 2017 at Nitte Meenakshi Institute of Technology, Bengaluru, India. His areas of interest in Automobile, Production