International Journal of Advanced Research Methodology in Engineering & Technology,
Volume 1, Issue1, December 2016

Computational Study on Optimization of Gas Turbine Combustion Chamber

Kannan.G[1], D.Daljit Majil[2], S.Guruprasaath[3], M.Karthik[4]

Vel Tech Dr.RR & Dr.SR Technical University, Chennai

Assistant Professor, Department of Aeronautical Engineering, kannang@veltechuniv.edu.in[1]

B.Tech Aeronautical Engineering, daljit.ae13@veltechuniv.edu.in[2]

B.Tech Aeronautical Engineering, gurulovesspace@gmail.com[3]

B.Tech Aeronautical Engineering, kkplaypoy721@gmail.com[4]

Abstract

Main objective of this work is to determine the optimum mass flow rate on fuel injector of a typical Gas turbine Engine .This paper presents the design of combustion chamber followed by three dimensional simulation to investigate the velocity and at the exit part of combustion chamber which is nothing but the entry region of Turbine section. Fuel Considered for the simulation process is methane (CH_4). CFD simulation has been made using ANSYS CFX 14.5 software to analyze the flow pattern within the combustion chamber. A 3D combustion chamber is designed with CATIA V5 which is later exported to ANSYS CFX 14.5

Nomenclature

m	Mass flow rate	kg/s
v	Velocity	m/s
D_o	Orifice Diameter	mm
L	Length	mm
A	Area	m^2
v_{ip}	Velocity of primary Air	m/s
V_{is}	Velocity of Secondary Air	m/s
V_e	Exit Velocity	m/s
P_e	Exit pressure	Pa
T	Temperature	K

Introduction

Combustion Chamber is one of the main components of an Aircraft Gas Turbine Engine. In Combustion chamber Fuel and Compressed air is mixed to appropriate ratio. Fuel is injected with Fuel Injector at specific mass flow rate. Can type Combustor is considered [2].In combustion Process takes place because of Thermodynamic Cycle called Isobaric Process, Which means that during the thermodynamic process Pressure will constant.

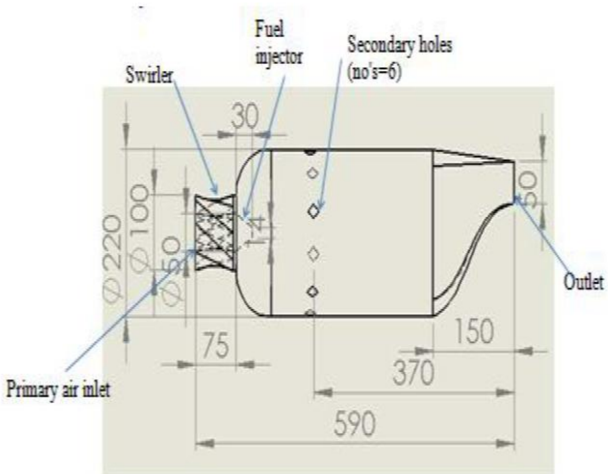
Computational Study

Experimental study is one of the methods to engineering problems, but this method is very costly. Hence this difficulty can be rectified by CFD.In the CFD problem is simulated to software and it proves for efficient tool and also analysis for various flow parameters.ANSYS CFX 14.5 software is used for simulation [1].

Combustion Chamber

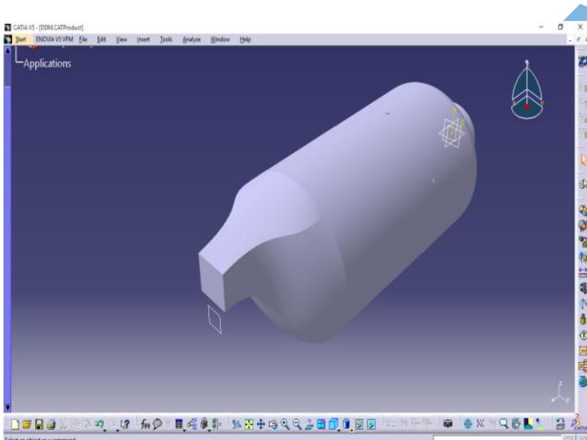
Geometrical parameters of Combustion Chamber os taken from [2]. Combustion chamber is having a length(L) of 590mm, Injector diameter(D) of 50mm and its orifice diameter of 4mm, A can is having diameter of 220mm and it consist of six holes for secondary air flow which will make a flame stabilzation having Area of 33.50mm^2. Primary Air inlet is having diameter of 100mm. Length of the injector is 105mm.A can combustor is having rectangular outlet area of 2.5m^2.

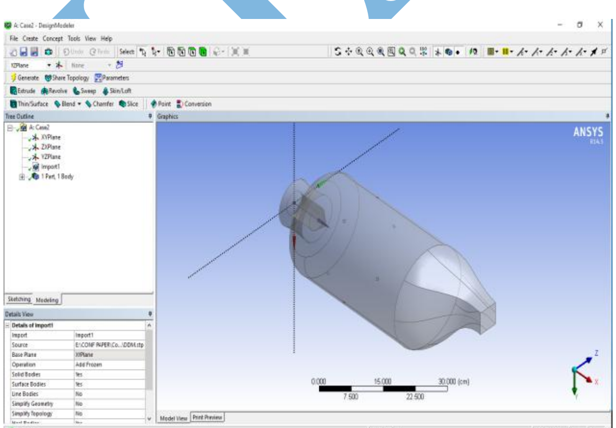
IJARMET



Modelling

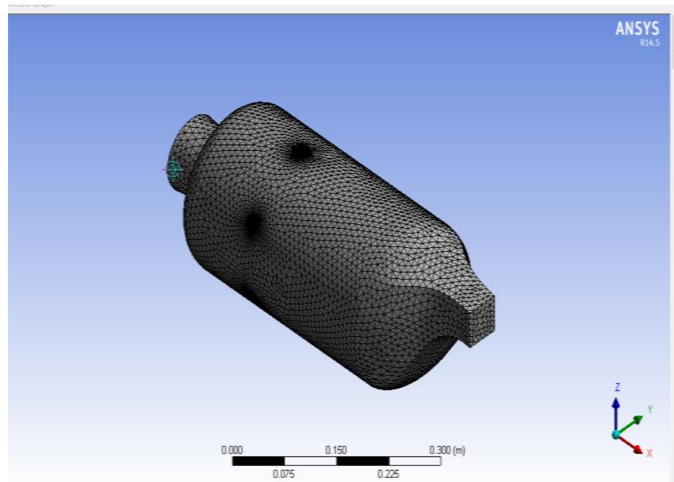
Modelling of 3D Can Combustor is done by CATIA V5 software with appropriate dimensions. Model is then saved to .stp format for export purpose.





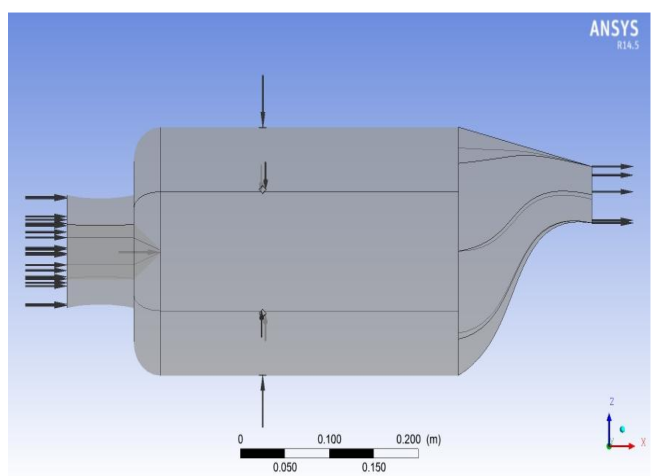
Meshing

3D model which is designed with CATIA V5 software is imported to ANSYS CFX 14.5 then it is meshed with fine mesh.



Boundary Conditions

Boundary Conditions for simulation of Can type Combustor is made with [2].Can combustor is consist of three inlets Primary inlet, Secondary Inlet, Fuel Injector. Primary Inlet is having velocity of 10m/s, Secondary Inlet is having Velocity of 6m/s and Fuel Injector injects fuel at mass flow rate inlet. Turbulence intensity of 10% and mass fraction of f=0. And Boundary condition for Fuel Inlet has been made with respect to properties Methane (CH_4) [1][2].



CFD Simulation

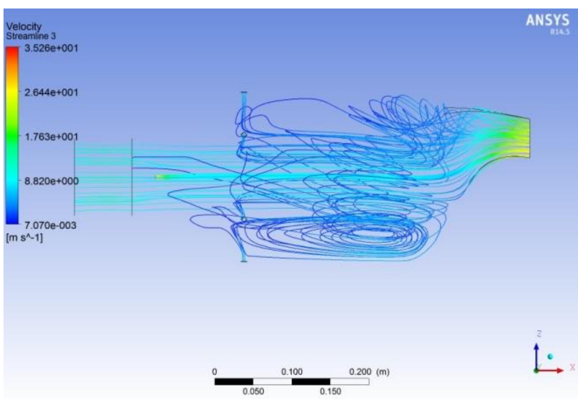
Simulation is carried for various cases with respect to various mass flow rate of Fuel Injector. Various Mass Flow rates [2] are given in following Table,

Cases	Mass Flow Rate (Kg/s)
1	0.0005
2	0.001
3	0.0015
4	0.002
5	0.0025
6	0.003

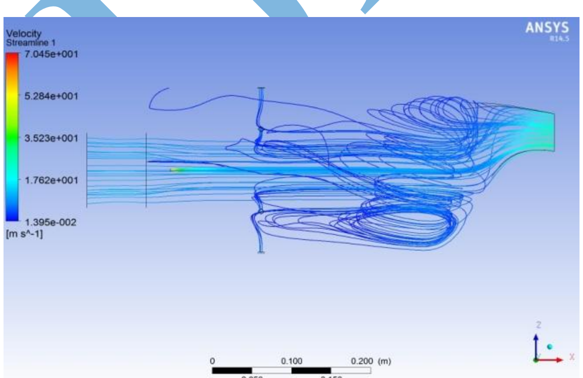
Table 1

Results and Discussion

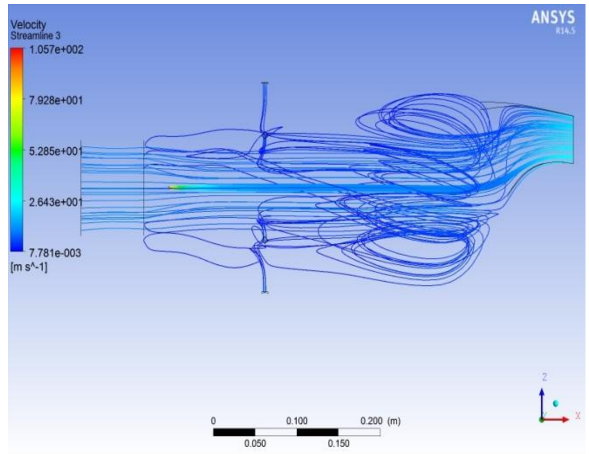
Streamline pattern has been made for various cases of simulation.



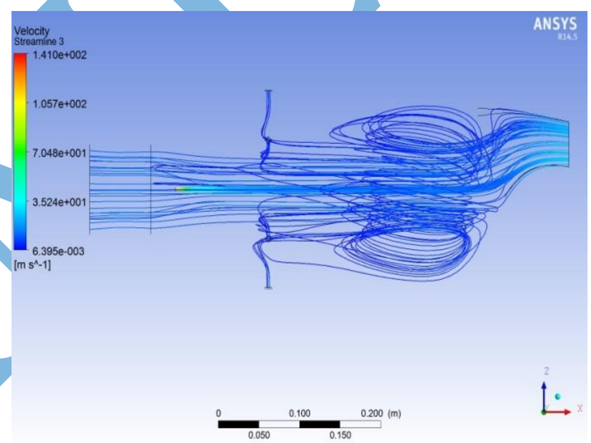
Case 1



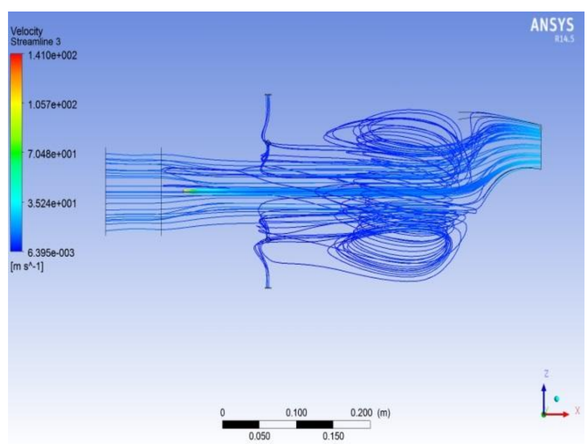
Case 2



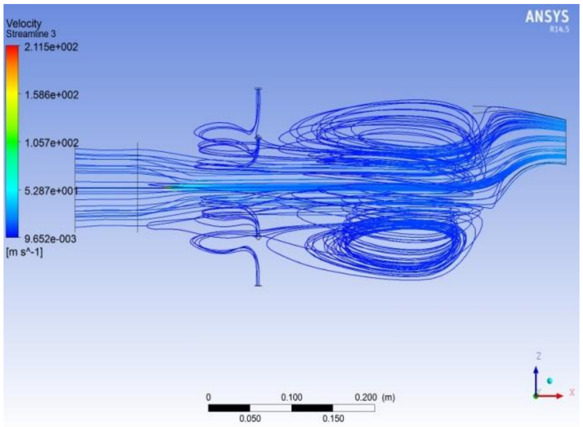
Case 3



Case 4



Case5



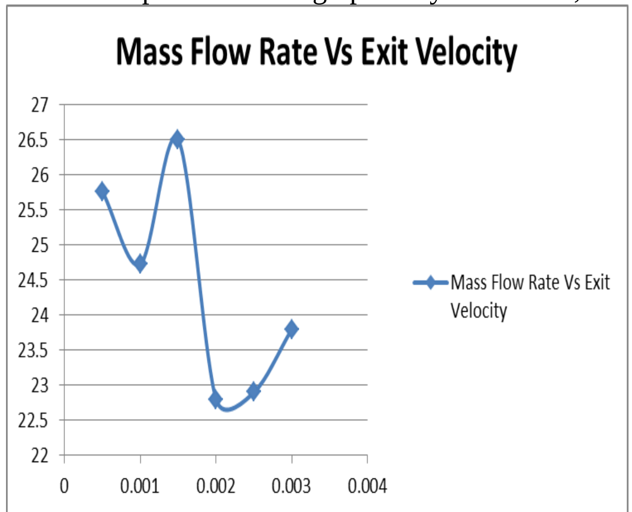
Case 6

Cases	Mass Flow Rate (Kg/s)	Velocity (m/s)
1	0.0005	25.76
2	0.001	24.73
3	0.0015	26.5
4	0.002	22.8
5	0.0025	22.9
6	0.003	23.8

Table 2

Graphical Representation

CFD simulation has been made for various cases mentioned above. Variation of exit velocity with variation of Mass Flow Rate has been represented graphically below,



Optimization of Mass Flow Rate

With the help of Table 2 and Graphical representation, Velocity value has to be more at exit of Combustion Chamber. So velocity at Case 3 is high which is having mass flow rate of 0.0015kg/s.

Hence Optimum mass flow rate for Can type combustor is 0.0015kg/s, which is mainly depends on orifice diameter of Fuel Injector.

Conclusion

By the CFD simulation of Can type Combustor, We would like to conclude that Orifice Diameter of Fuel Injector has to be designed such a manner that it should inject the fuel at optimum Mass Flow Rate.

References

[1] P.Sravan Kumar, P.Punna Rao; **Design and Analysis of Gas Turbine Combustion Chamber;** International Journal Of Computational Engineering Research (ijceronline.com) Vol. 03 Issue. 12.

[2] Firoj H Pathan, Nikul K Patel, Mihir V Tadvi; **Numerical Investigation of the Combustion of Methane Air Mixture in gas Turbine Can-Type Combustion Chamber**; International Journal of Scientific & Engineering Research, Volume 3, Issue 10, October-2012 1 ISSN 2229-5518.

[3] Gas Turbine by V.Ganesan.

[4]https://en.wikipedia.org/wiki/Combustion_chamber

[5] https://en.wikipedia.org/wiki/Combustor

IJARMET