Large Eddy Simulation of temperature distribution in an aero engine annular combustor with a swirler and Nanoparticle injection

[1]Vishnu Sasidharan, [2]Arun Kumar A.R

Assistant professor
Department of Aeronautical Engineering
Mount Zion College of Engineering, Pathanamthitta

Abstract - Computational fluid dynamics (CFD) approach can reduce the expenses as well as time to provide an insight into the characteristics of flow and combustion process inside the combustion chamber at design stage. Geometry of the aero engine combustor simulated for investigation is an 18° sector of an annular combustor. Primary secondary and dilution holes are simulated on the inner and outer liner walls. Swirlers are provided on the thrust chamber to give uniform rotational velocity distribution as well as a thicker and higher velocity reverse flow core. Temperature has been analyzed in the annulus region. Liner holes provided on the surface of the chamber allows an air lining to form around the inner surface of the combustor, thereby maintaining the inner temperature of the combustor by reducing the heat flow towards the outer casing. So the location and orientation of the liner holes are selected on the basis of minimal heat loss to the surroundings and heating of the outer casing so that the material selection for the casing and related areas become less complicated and can reduce the application of various wall cooling methods. Nano fuel particle injection is employed to ensure thorough mixing and complete combustion of fuel there by reducing the pollutant emissions.

Introduction

Gas turbine engines have been used over the years as a useful source for vehicle propulsion and power generation. Air enters the engine through a compressor which increases the air pressure to achieve the proper conditions for combustion. The compressed air enters the combustion chamber where fuel is added and burned. The hot gasses from the combustor then pass through the turbine section of the engine, which is connected to a central shaft that links the compressor and turbine sections. A portion of the momentum from the hot combustion gasses is harnessed by the turbine blades to rotate the shaft, which resultantly turns the compressor and runs the engine.

One major topic of gas turbine research is on viable methods of improving fuel efficiency. It is known that a basic way to increase efficiency is to spin the engine faster. In order to do this, a higher burn temperature in the combustor must be achieved. Higher combustion temperatures, however, will produce larger amounts of pollutants. With the strict limitations of the modern day emission standards and the continuing desire for higher fuel efficiency in the gas turbine industry, it has become a growing importance for combustor design engineers to better understand the flow distribution within the combustor and the heat transfer process between the gas flow path and the combustor liner wall. With burn temperatures reaching levels higher than the melting point of the materials used to construct the combustors, the walls of the combustor must be cooled in order to prevent damage or failure. A detailed analysis of the combustor liner heat transfer will determine the necessary amount and location of cooling along the wall and will prevent any overcooled or undercooled regions. This will allow the combustor to run at an optimal temperature for emissions and efficiency with minimal losses to the combustor liner.

A combustion chamber is the most heavily thermal load part in an aero engine, in which both ignition and combustion keep going on, and normally whose service life is very short. Since very limit data can be obtained from expensive engine tests due to its serious work environment, CFD is often employed to simulate the complex physical processes in a chamber. So far there are very plenty of available researches to validate the reliability of CFD analyses for investigating the flow, combustion and heat transfer in a combustor. As powerful computing technologies are continuously and rapidly improved, the feasibility of using CFD analyses is undoubted for combustor design. Many researchers employed CFD analysis to study their acting flow within the combustor tube only. Pratt and Whitney Corporation developed a CFD analysis on an entirePW6000 combustor domain to predict temperature distribution at the combustor exit and compared the CFD results with the full annular rig-test data. However, all above-mentioned studies on ''full combustor'' omitted cooling devices or simplify cooling holes to slots. The demand for modern gas turbines with higher thermal efficiencies has greatly prompted designers to increase the turbine inlet gas temperatures steadily. One thousand and nine hundred Kelvin is a typical combustor inlet temperature for current aero-engine designs, which is well above the sustaining temperature of the tube material. One method of protecting combustor tube in these harsh environments is discrete holes film cooling. The cooling air/fluid is injected through small discrete holes into the tube internal boundary layer, forming a protective film on the surface. Film cooling is widely used in the modern gas turbine combustors to cool and protect tube walls and was extensively studied in last 30 years. However, most of the researches in the open literature just concentrated on flat or curve plates with film injection through slots or rows of cylindrical holes, very few consider the integration of combustor and full film cooling.

Model description.

Model used in the analysis is a sector of an annular combustor, an 18 sector designed in CATIA. The model consists of inner and outer flame tubes provided with liner holes, for secondary and dilution air an combustor assembly consists of many different elements/parts and small design details, such as injecting devices, mounting brackets and flanges or buckets on the holes. Because the mass flow distribution and oil profile can be obtained from an engine model test in AEC, some attached parts and small details were cleaned up if they do not significantly affecting the fluid flow, combustion and heat transfer performance in the combustor. The geometrical clean-up is necessary for simplifying the processes of 3D model creation and mesh generation. Figs. below show the 3D geometry of the combustor after the model clean-up and simplification. The combustor is seasonal in geometry and there are 20 nose intakes. So, the simulative domain adopted here is 1/20 of the whole combustor.

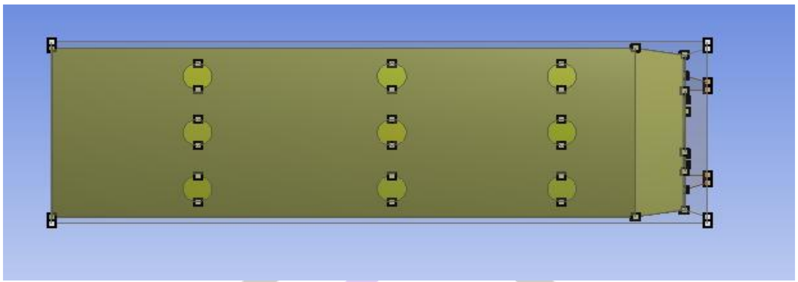

Fig: 1 liner holes

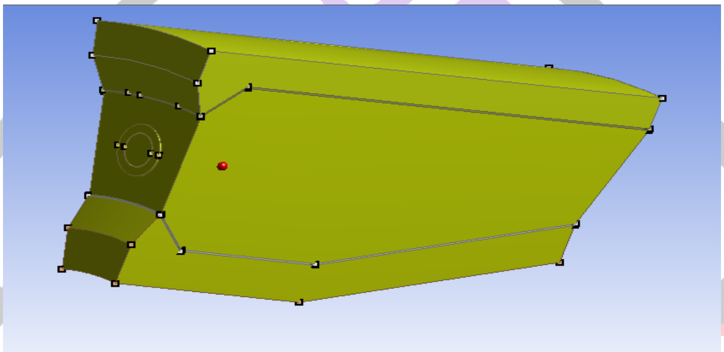

Fig: 2 3D Geometry model

Boundary conditions and flux distribution

The boundary conditions illustrated are summarized in Table 1. The air entering all inlets had a temperature of 860 K and a static pressure of 3.36 MPa. The fuel temperature was 433 K injected from the surface at the pressure of 3.16 MPa. The mass flow fluxes were also defined according to the test results inAEC, China and are listed in Table 1Atomization of liquid fuel was considered to obey Rosin–Rammler law with average drop diameter _d ¼ 30 lm and non-uniform exponent n = 2.5. The side wall of the fluid zone was set as rotating periodical boundary condition

Table: 1 Boundary condition and flux distribution

BOUNDARY NAME	BOUNDARY CONDITION	FLUX (kgs^{-1})	TOTAL FLUX (kgs^{-1})
Outer hoop air inlet	Mass flux	1.16215	3.1971
Inner hoop air inlet	Mass flux	1.2052	
Nose coolingair inlet	Mass flux	0.32065	
Fuel and first rotational air inlet	Velocity distribution	0.5091	
Second rotational air inlet	Velocity distribution	0.156	
Outer hoop air outlet	Mass flux	0.0936	3.1971
Inner hoop air outlet	Mass flux	0.14	
Chamber tube outlet	Avg. back pressure	2.8075	

CFD solver

CFD solver used for the analysis is ANSYS14.0. The geometry developed in accordance with the actual combustor model in CATIA. The dimensions are taken as per the model and imported in ANSYS, there the geometry was subjected to certain reconstructions to ensure proper meshing in the pre meshing segment of ANSYS the resultant geometry had undergone hexa mesh with about 3 billion cells. The solid surfaces were made as walls and given boundary condition as wall, the fuel and air inlet holes as mass flow inlet. And an average back pressure is assumed in the chamber tube outlet. The model is imported to the FLUENT for further analysis. The fuel used in the combustion process is kerosene. Which is injected at a temperature of 433K rosin rammler injection technique is employed. The fuel particle is injected as Nano particles i.e., the size of the fuel particles are of the range of 10^{-9}The heat release model is given by the following two-step scheme, which allows for calculation of CO and unburned hydrocarbons (finite rate kinetics):

$$C_xH_y + (x/2+y/4)O_2 \rightarrow xCO + 0.5yH_2O$$

$$2CO + O_2 \rightarrow 2CO_2$$

Results and discussions

The combustor modeled is first analyzed for temperature distribution and flow pattern without employing swirlers and Nano fuel injection and the following results obtained. The below results shows that the proper mixing is not obtained and the temperature distribution is uneven. The same combustor is analyzed by using a swirler and the fuel is now injected as nano particles and the results are more satisfactory.

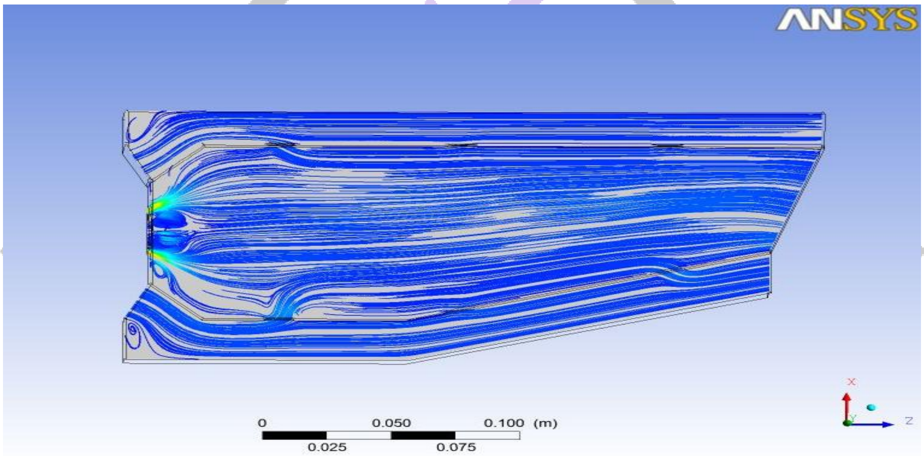

Fig: 3 flow pattern

From the below table it is visible that among the 1000 particles tracked for analysis only 749 particles are evaporated this had increased drastically during the implementation of a swirler, where it has increased to 960 and even raised after Nano injection of fuel. This shows that proper mixing of particles takes place when the fuel particles are injected as Nano particles and the turbulence is increased by the swirler

Table: 2 Fuel particle statistics

	WITH OUT SWIRLER	WITH SWIRLER	WITH SWIRLER AND NANO INJECTION
PARTICLES TRACKED	1000	1000	1000
PARTICLES EVAPORATED	749	960	969
PARTICLES ESCAPED	210	28	24

INCOMPLETE	41	12	07

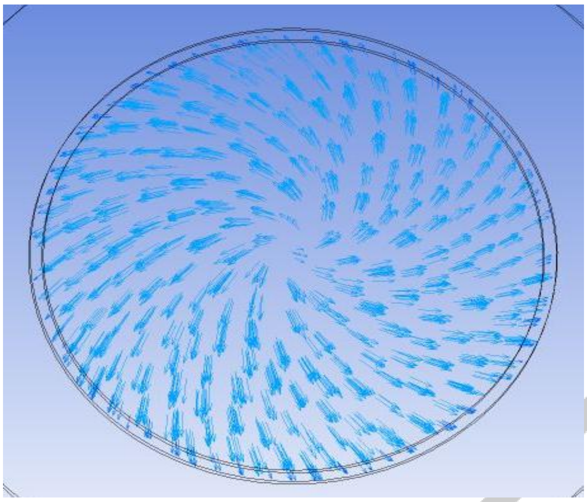

Fig: 4 Rotational air

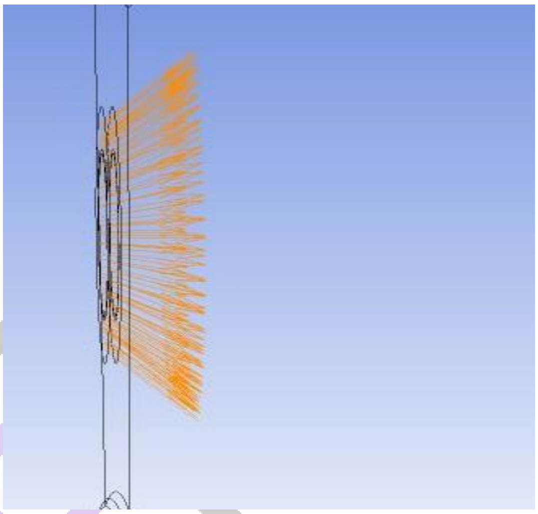

fig: 5 Fuel spray pattern

The inlet air with the application of swirler enters in anticlockwise direction which ensuresthorough mixing of fuel particles with air. The fuel is injected as surface cone spray at a cone angle of 10°

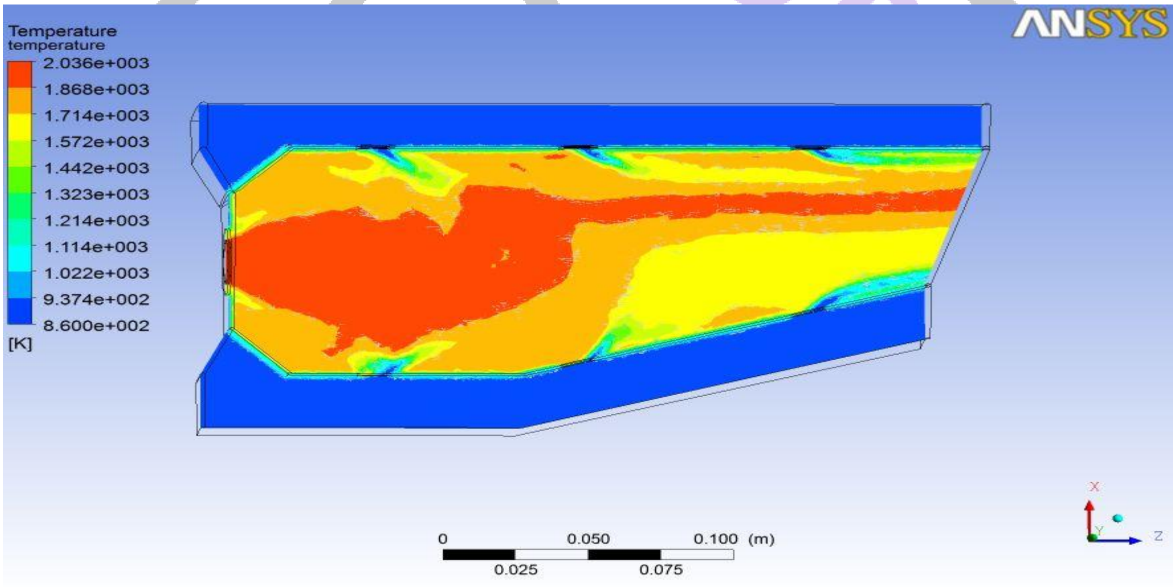

Fig: 6 Temperature distributions without swirler

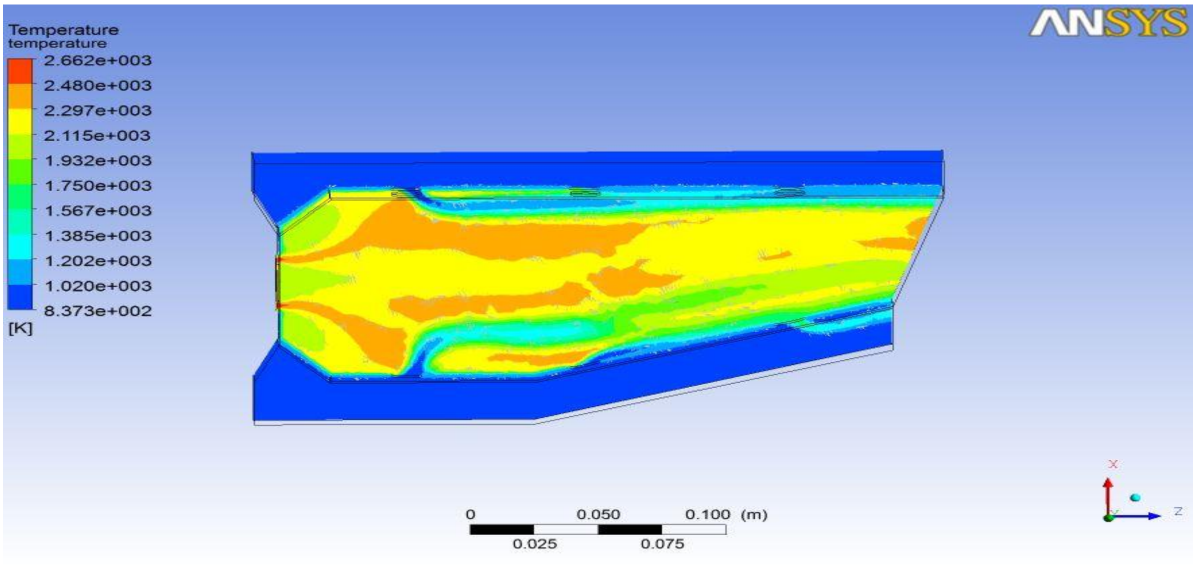

Fig: 7 Temperature distributions with swirler

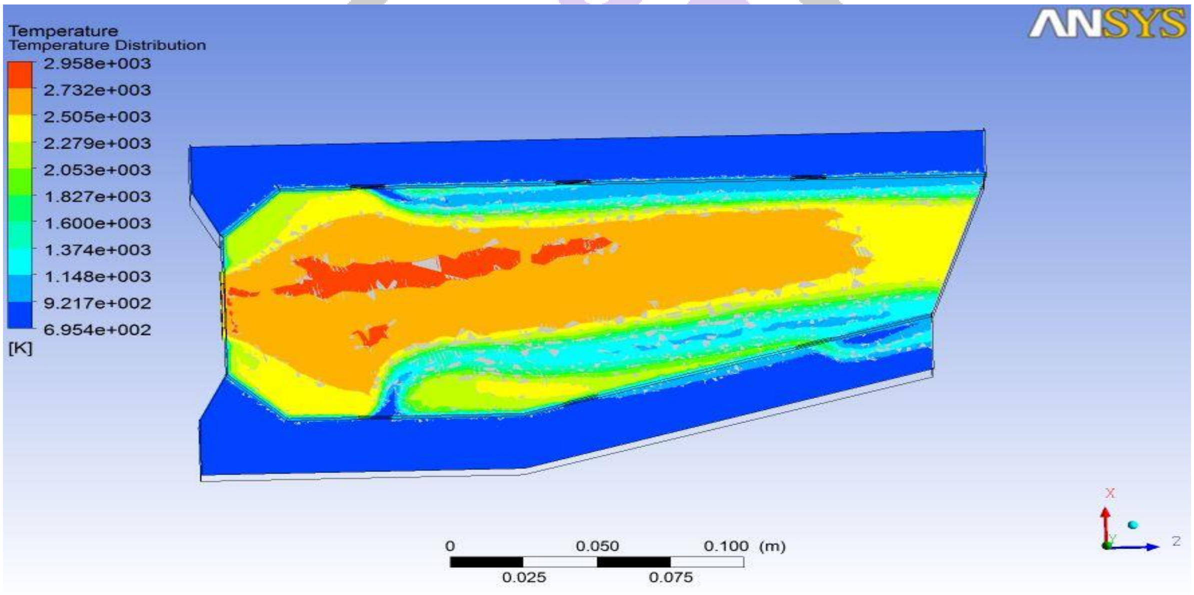

Fig: 8 Temperature distributions with swirler and Nano injection

The temperatures analyzed during the process have a large extend of variation when employed with a swirler and even satisfactory when the fuel particles are injected as Nano sized. The below table shows the maximum and minimum value of temperature at the three analyzedconditions. And the available temperature when the combustor is having a swirler and the application of nano particle injection is around 3000K and the temperature at the casing is reduced to about 700K.

Table: 3 Temperature distributions

TEMPERATURE (K)	WITH OUT SWIRLER	WITH SWIRLER	WITH SWIRLER AND NANO INJECTION
MAXIMUM	2036	2662	2958
MINIMUM	860	837	695

Conclusion

From the above results we can easily conclude that the temperature distribution is satisfactory with swirlers and improved injection technique of Nano particle injection. The thereby we can easily say that the presence of turbulence inside the combustor provides thorough mixing of particles and pollutant emissions can be reduced. Since the incomplete burnt particles are reduced during the presence of turbulence and newer injection technique. The maximum available temperature is observed to be increased and the heat convicted to the outer casing is reduced simultaneously

References

[1] Combustion and cooling performance
[2] in an aero-engine annular combustor
L. Li a, X.F. Peng ,, T. Liu*
Laboratory of Phase change and InterfacialTransport Phenomena,
Department of Thermal Engineering,
Tsinghua University, Beijing, China
[3] Heat transfer characteristics of an impinging Premixed annular flame jet
H.S. Zhen, C.W. Leung, C.S. Cheung*
[4] Evaluation of an experimental short-length annular combustor: one-side-entry dilution airflow concept
Francis M. Humenik and James A. Biaglow
[5] Performance of an annular combustordesignedfor a low-cost turbojet engine *James S. Fear*
[6] Heat transfer and flow measurements in gas turbine engine can and annular combustors *Andrew C. Carmack*