Scientific Journal of Impact Factor (SJIF): 3.134
ISSN (Online): 2348-4470
ISSN (Print) : 2348-6406

International Journal of Advance Engineering and Research Development

Volume 2, Issue 4, April -2015

COLD FLOW ANALYSIS IN AN ANNULAR GAS TURBINE COMBUSTOR USING CFD TOOL

Parveen Kumar Saini[1], Gurpreet Singh Malhi[2] Gurkirat Singh[3]

[1] *Assistant Professor, Mechanical Engineering Deptt., Chandigarh Engineering College, Landran (Mohali).*
[2] *Assistant Professor, Mechanical Engineering Deptt., Chandigarh Engineering College, Landran (Mohali).*
[3] *Assistant Professor, Mechanical Engineering Deptt., Chandigarh Engineering College, Landran (Mohali).*

ABSTRACT: *In continuous devices such as a gas turbine combustor, there exists a range of complex interacting physics, included are vortex flows with turbulent effect. Rigorous descriptions of these phenomena are, however, either not available or require mathematical models which are complex for computation, when taken together in the context of multi-dimensional flows. For these reasons, models of varying degrees of sophistication in the model are continuously increasing with improvements in numerical methods, computer capabilities and physical understanding. This paper focused on the velocity profiles, flow distribution and pressure drop in an annular gas turbine combustor. Flow-split through different liner holes and swirler has been estimated. It is seen that percentage flow-split through the outer liner surface holes and inner liner surface holes can be varied by changing the diameter of the liner holes even though the inlet velocity has been maintained constant. Various parameters like pressure loss, velocity, mass flow distribution, Mach number, and speed at various locations, have been estimated. In addition, the effect of varying the primary inner holes on the cold flow performance of the combustor has been studied. A 18^0 sector of the combustor has been modeled in Pro-E Wildfire 4.0 and cold flow analysis has been carried out using the software CFD tool 10.*

Keywords*: Annular gas turbine combustor, Mach number, Computational fluid Dynamics, Finite element analysis*

I. INTRODUCTION

Designing a combustion chamber for a gas turbine engine requires expensive testing with much iteration. Previously, the designer relied primarily on the past experience, test results and analysis based on empirical formulations to make the final design decisions. High temperatures, pressures and flow rates in the combustor result in a flow field where comprehensive experimental data are so expensive to obtain. Thus, Computational Fluid Dynamics (CFD) is an attractive design tool, since it has the potential to explain the flow physics inside the combustor. A numerical analysis can be used to reduce the number of design iterations by providing an insight into the changes that a design parameter should undergo regarding the characteristics of the flow. Improving the prediction capabilities and reliability of CFD methods will reduce the cycle time between idea and a working product.

In this article, we study a model gas turbine combustor. The entire flow field is modeled, from the compressor diffuser to turbine inlet. In order to obtain the accurate result with less time and cost, we modeled an 18 deg sector of the combustor using Pro-E Wildfire 4.0. Then it is imported into the software CFDesign 10. After meshing it, we get the finite element model, which is used for fluid analysis. On the base of analysis result it can display the pressure contour and the velocity vector.

II. GEOMETRICAL MODEL

A sectional view of a model annular gas turbine combustor model is as shown in fig.1 the geometry is simplified because of limitation of the computational capabilities. But an attempt is made to retain most of the necessary features, which were necessary to understand the complex flow physical behavior in a gas turbine combustor. A 3D model of annular gas turbine combustor is also shown in fig.2.

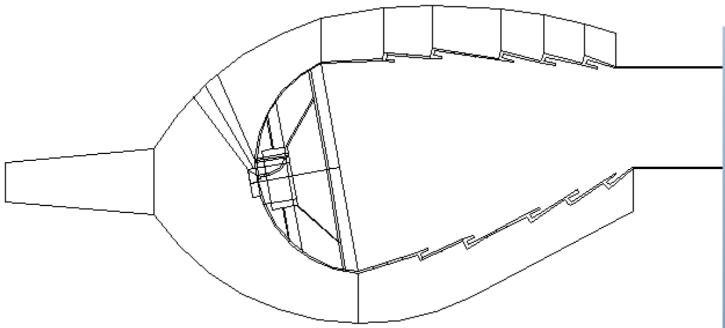

Fig.1 Sectional view of annular gas turbine combustor

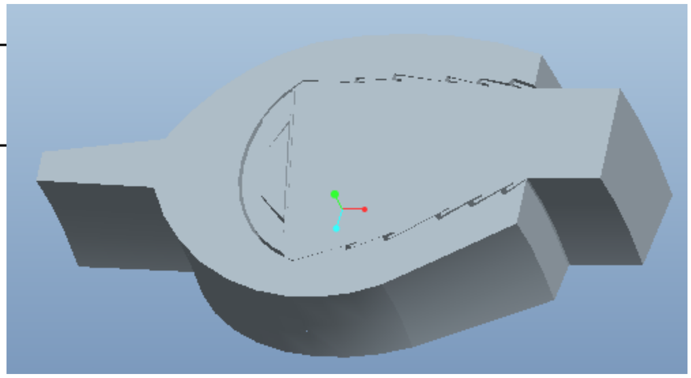

Fig. 2 Assembled model of combustor in 3-D

III. FINITE ELEMENT MODEL

Generated model is then imported in CFDesign 10. After meshing it, we get the finite element model (see fig.3), which is used for fluid analysis. CFDesign performs a comprehensive topological interrogation of the analysis geometry and determines the mesh size and distribution on every edge, surface, and volume in the model.

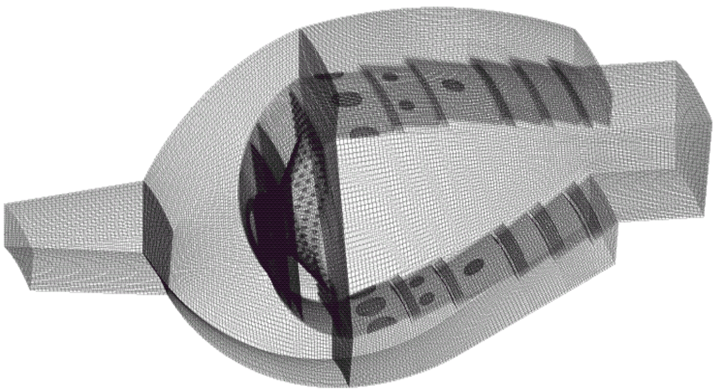

Fig. 3 Meshed model of combustor

BOUNDARY CONDITIONS

The boundary conditions are as follows:

- At inlet of the combustor mass flow rate is 4.7422 kg/s.
- Viscosity of fluid is 3.5E-05kg/m-s.

IV. CALCULATION RESULTS AND ANALYSIS

3D constant incompressible viscid N-S equation is used as governing equation and standard model is applied to simulate turbulent computation. Fig.4, 5 and 6 show the pressure contour and the velocity vector on the symmetric section of the model, respectively.

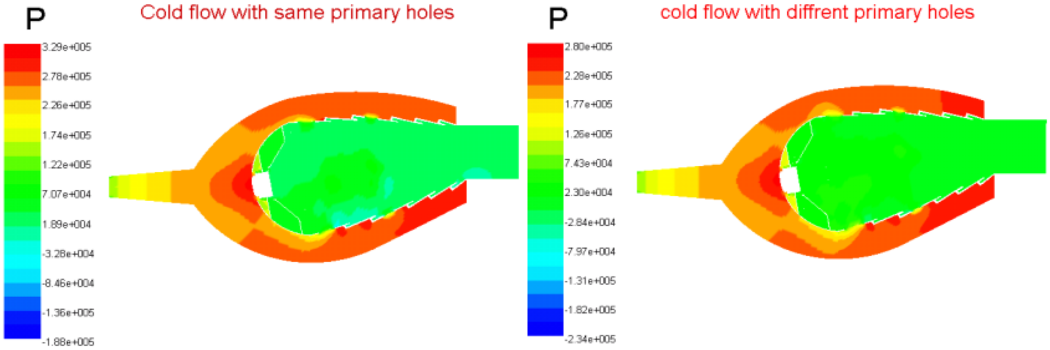

Fig.4 Pressure contours

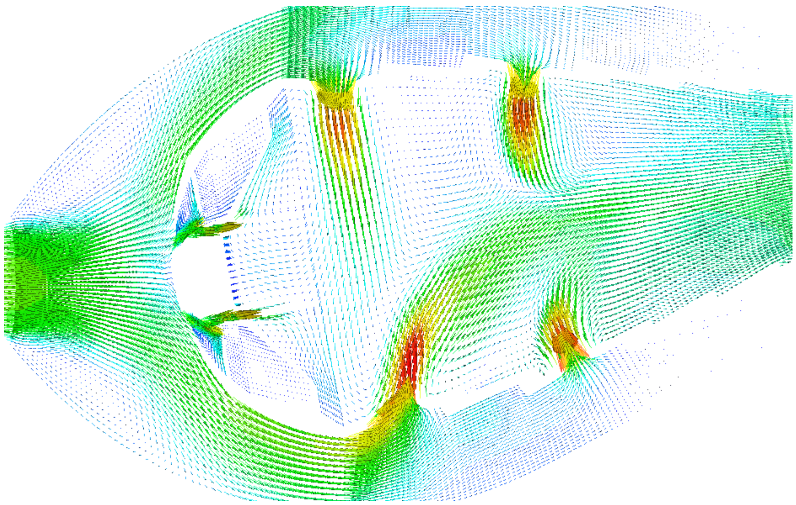

Fig.5 Velocity vector plots for same primary holes

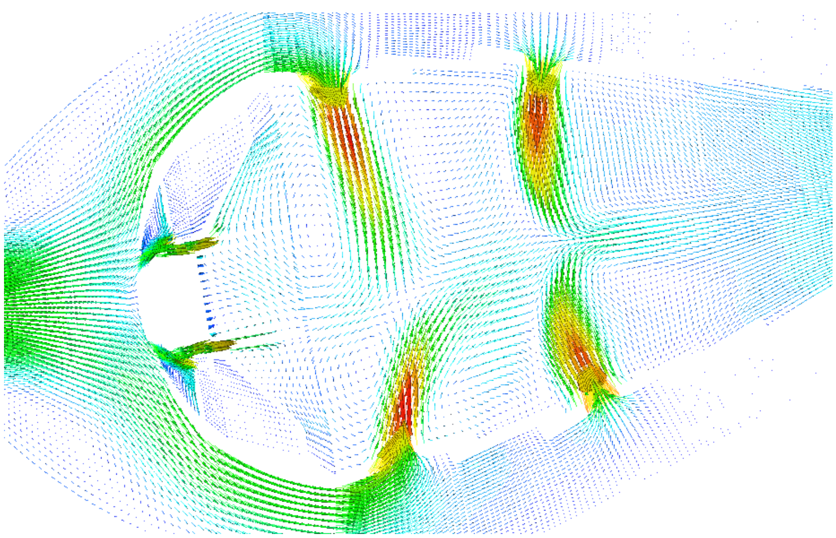

Fig.6 Velocity vector plots for different primary holes

The mass splits inside the combustor is given in the table 1 and the effective pressure loss at various locations by changing the inner annuls liner holes are compared in table 2.

TABLE 1 – Percentage of Mass Flow inside the Combustor

Percentage of mass flow inside the combustor		
Name	**For 20 mm holes diameter**	**For 24mm hole diameters**
Inlet	1.00E+02	1.00E+02
Inner Annulus Inlet	4.24E+01	4.69E+01
Outer Annulus Inlet	4.23E+01	3.90E+01
Cowl	1.53E+01	1.41E+01
Swirler	1.18E+01	1.08E+01
Dome holes	3.48E+00	3.25E+00
Flare Holes	3.48E+00	3.25E+00
Inner Annulus Primary	2.75E+01	3.30E+01
Inner Annulus Secondary	7.48E+00	6.92E+00
Inner Annul Dilution	7.42E+00	6.98E+00
Outer Primary Hole	2.76E+01	2.55E+01
Outer Annulus Secondary	7.33E+00	6.70E+00
Outer Annulus Dilution	7.37E+00	6.84E+00
Exit	1.00E+02	1.00E+02

TABLE 2 - Pressure Loss at Various Locations in Combustor

Pressure loss compression		
Regions	**%Pressure Loss with same liner holes**	**%Pressure Loss with different liner holes**
Outlet of pre-dump	7.66E-03	8.39E-03
Cowl Outlet	9.83E-02	8.87E-02
Swirler Outlet	3.59E+00	1.39E+00
Dome Holes Outlet	5.53E-01	4.96E-01
Flare Holes Outlet	5.39E-03	4.03E-03
Primary Zone Exit	8.26E+00	6.88E+00
Secondary Zone Exit	8.49E+00	7.31E+00
Dilution Exit	8.72E+00	7.60E+00
Combustor Exit	8.90E+00	7.71E+00

V. CONCLUSION

The conclusions drawn from the study are given below:

- The vector plots and contour show the characteristic recirculation pattern at various locations in the combustor.
- The total pressure loss inside the combustor is found to be 7.71%.
- Mass splits at different regions of the combustor are obtained from simulation results and are tabulated.
- Flow-split through different liner holes has been estimated. It is seen that percentage flow-split through the outer liner surface holes and inner liner surface holes can be varied by changing the diameter of the liner holes even though the inlet velocity is been maintained constant.
- The pressure loss inside the combustor can be decreased by just varying the liner holes and pressure loss variation occurred for this case is 1.19%.

REFERENCES

[1]. User's Guide CFDesign®V10 Upfront CFD", Blue Ridge Numerics, Inc. 1992-2009
[2]. CFDesign Technical Reference Upfront CFD Version 9.0", Blue Ridge Numerics, Inc. 1992-2006
[3]. Jiyuan Tu, Guan Heng Yeoh, Chaoqun Liu, "Computational Fluid Dynamics: A Practical Approach", ISBN - 978-0-7506-8563-4, Butterworth-Heinemann (2008)
[4]. S N Singh, V Seshadri, R K Singh and T Mishra, "Flow Characteristics of an Annular Gas Turbine Combustor Model For Reacting Flows Using CFD", Journal of Scientific and Industrial Research, vol.-65, pp. 921-934, (2006)
[5]. S.M. Correa and W. Shyy, "Computational models and methods for continuous gaseous turbulent combustion", Progress in Energy and Combustion Science, vol.- 13, no.- 4, pp. 249-292 (1987)
[6]. W. Shyy, M.E. Braaten and D.L. Burrus, "Study of three-dimensional gas-turbine combustor flows", International Journal of Heat and Mass Transfer, vol.- 32, no.- 6, pp.1155-1164, (1989)

