ISSN (e): 2250 – 3005 || Volume, 08 || Issue, 8|| August – 2018 ||
International Journal of Computational Engineering Research (IJCER)

No Reaction Flow Analysis of Aero-Derivative Annular Combustor of a Turbo Shaft Engine

Raja Marudhappan[1], Udayagiri Chandrasekhar[2], K Hemachandra Reddy[3]
[1]Research Scholar, JNTUA, Ananthapuramu - 515 002, India
[2]Vel Tech University, Chennai- 600 062, India
[3]JNTUA, Ananthapuramu – 515002, India
Corresponding Author: Raja Marudhappan

ABSTRACT
Two different configurations of annular combustor of a generic 1100 kW class aero-derivative gas turbine are subjected to steady state incompressible cold flow analysis. One model has combustion liner with plain holes and the other with plain holes in recirculation and primary zones and dilution tubes in dilution zone alone. The simplified computational domain is discretized with Polyhedral cells. The two equation k-Ɛ model is applied for simulating the turbulence with standard wall function. The Reynolds Averaged Navier-Stokes (RANS) equations are solved over the discretized domain using pressure based Semi-Implicit Method for Pressure Linked Equations(SIMPLE) and finite volume method. The calculated velocity fields at combustor exit, pressure contour and Streamlines on Meridional plane are presented. The generation of turbulent kinetic energy and its dissipation are presented and discussed. Though the experimental data on similar combustor models are not available in literature, the overall aerodynamic performance of combustors resembles typical proven annular combustor aerodynamics.
KEYWORDS: Annular combustor, CFD, Turbo-shaft, Flow analysis, Turbulence, Streamlines, RANS

Date of Submission: 20-07-2018 Date of acceptance: 04-08-2018

I. INTRODUCTION

The overall performance of a combustion chamber depends on proper supply of combustion air at right place, direction and quantity. The combustion air needs to be apportioned effectively to primary zone combustion to stabilize the flame, in secondary zone to maximize the burning rate and in the dilution zone for proper temperature reduction. The stabilization of flame near the injector solely depends on the formation and sustaining of recirculation zone around the injector. The major objectives of a combustion system is to reduce the velocity of incoming air and distribute to different zones to maintain uniform flow without parasitic losses to obtain maximum burning rate and desired pattern factor at the exit.

The design of aero derivative combustion chamber is a challenging task to the researchers since the space available for combustion is very small. The researchers need to carefully assess proper mixing of incoming air with the hot gas to ensure flame stability and to reduce the pollutant emission and within acceptable noise level. The provisioning of several features considering aerodynamic and combustion requirements makes the combustor geometry more complex. The successful design of a combustion chamber requires the knowledge of flow recirculation, jet penetration and mixing. A good mixing is required in the primary zone for achieving high combustion rates by maintaining the emission meeting the statutory requirements. The dilution zone must provide proper jet penetration in order to supply hot gas to the turbine inlet with acceptable pattern factor. The penetration of dilution jet is improved by providing dilution tubes.

This study focusses on the flow analysis on two different combustor configurations, one without dilution tubes and another with dilution tubes. A generic annular combustion chamber of a 1100 kW class aero derivative gas turbine is considered for the flow analysis. One configuration has combustion liner with plain holes and the other configuration has liner with extended tubes only in the dilution zone. The objective of this research is to perform cold flow analysis in a realistic annular combustor using commercial CFD code to obtain aerodynamic aspects of the flow at the design inlet pressure.

II. LITERATURE REVIEW

The early combustion research dates back to 1941 when the first British turbojet engine powered flight was made [1]. Since then the combustion system has undergone several structural and material changes to meet the ever demanding aerodynamic and combustion requirements. The continuous demand from the customers for enhanced performance at varying altitudes necessitated introduction of many features thus making the combustor geometry more complex. The design of an aero derivative combustion system requires detailed understanding of flow phenomenon inside the combustor. The development of any new combustor begins with the modeling and thorough understanding of aerodynamics in the first phase before going for a full-fledged combustion modeling and experimental validation.

Three dimensional unsteady turbulent flow analysis on an annular combustor was performed using the computer program called CONCERT[2]. The analysis used standard two equations k-e turbulence model. The multi-step predictor-corrector method is followed for pressure correction equation. The calculated velocity at combustor exit plane is presented. The three dimensional streamlines from main inlet and from dilution holes are also presented. The study suggested that the modified CONCERT algorithm is capable of predicting the flow and combustion phenomena.

Turbulent isothermal fluid flow analysis was performed in a three dimensional can combustor[3]. Like many other similar researches this also used standard k-e turbulence model and pressure based SIMPLE solution algorithm. The non-staggered solution algorithm was used to close coupling between pressure and convective velocity. The velocity vectors and turbulent kinetic energy profile are presented. The research reported that the steep gradients in the flow are not correctly predicted and also suggested further refinement of grid.

A two dimensional axi-symmetric turbulent steady state flow analysis of a model can combustor was performed to assess suitability of numerical model [4]. The model used Renormalized Group (RNG) k-e turbulence model. The commercial CFD code Fluent was used for obtaining the solution of unsteady incompressible Navier-Stokes equations using SIMPLE solution algorithm.The CFD predictions were compared with experimental observations and they are found to be satisfactory.

The numerical analysis of aerodynamics and hydraulic loss were performed in a GTE-150 combustion chamber of a gas turbine power plant[5]. The analysis was done using commercial CFD codes. The system of Reynolds Averaged Navier-Stokes (RANS) equations were solved by finite volume method. The two equation k-e turbulence model with standard wall function was used. The analysis was done on compressible and incompressible flow model. The SIMPLE-Consistent (SIMPLEC) pressure correction algorithm for compressible flow and SIMPLE algorithm for incompressible flow were used. The calculated total pressure losses are compared to experimental data and are found in good agreement.

The steady state aerodynamic analysis is done on a model combustion chamber by applying three types of turbulence models [6].The three different turbulence models, viz. k-e, RNG k-e and Shear Stress Transport (SST) are applied to validate the suitability of turbulence model over the experimental predictions. The velocity and pressure variations at different combustor locations are presented. The predictions with SST turbulence model is found more satisfactory than other models.

A three dimensional numerical analysis of aerodynamic flow in a micro gas turbine combustor is performed on successively refined grids [7].The solution of RANS equations shows large total pressure losses in combustor because of combustor's non-conformance to proper aerodynamic requirements. The complex aerodynamic challenges in the design of an aero engine combustor are very much essential to meet the challenging future needs on both emission and performance requirements[8]. The importance of aerodynamic process in achieving optimized combustion efficiency, operational stability and combustor life over the engine life are illustrated. The effect of high swirl and high injector mass flow on the engine performance pose the threat of thermo-acoustic instabilities and strong compressor/combustor and combustor/turbine coupling.

The combustion aerodynamics of burners of Alstom SGT 600 gas turbine was numerically modeled to investigate the flow field and combustion properties of main fuel holes[9]. The unsteady solution on the discretized control volume was obtained by using open source CFD code OpenFOAM with k-e turbulence model. The simulated temperature field and experimental data are found to be within 0.1%. However the type of fuel used is not mentioned. The flow inside the swirl cups of gas turbine combustor is modeled to study the flow characteristics in a two dimensional axi-symmetric model [10].The Realizable k-e turbulence model with pressure based SIMPLE solution algorithm was used for solving the steady state incompressible RANS system of equations. Six swirl cups with different configurations were analysed. The analysis highlights the impact of design parameters on the flow field.

III. COMBUSTION CHAMBER

The schematic drawing of the combustion chamber with basic details are shown in Fig 1.0. The three dimensional models of the two types of combustor considered for flow analysis are shown in Fig 2.0 and Fig 3.0.The annular combustion chamber has become an ideal choice for aero gas turbine applications because of its clear aerodynamic layout. The geometry can be made very compact with lower pressure loss when compared to other types of combustion chambers. The combustion air delivered from the compressor enters combustion chamber through the annular space at the combustor inlet.Approximately 80 percent of combustion air enters the annular space. The remaining 20 percent of combustion air enters the primary zone through the swirler and doom.The combustor has no pre diffuser. The combustion chamber has outer combustion liner, inner liner and casings. The liners are provided with holes for proper apportioning of the combustion air and dilution air. The liners are subjected to highest temperature and hence proper cooling is important in structural and thermodynamic points of view. The liners are provided with multi jet impingement combined with film cooling arrangement to maintain liner temperature within acceptable level. The typical apportioning of combustion air inside the combustor is shown in Fig 4.0.

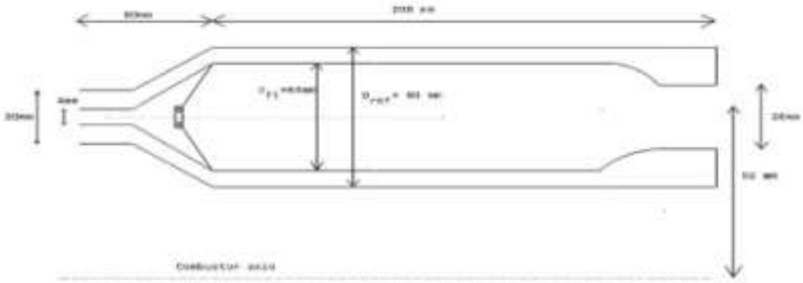

Fig 1.0 Schematics of basic combustion chamber

Fig 2.0 Combustion chamber without dilution tubes

Fig 3.0 Combustion chamber with dilution tubes

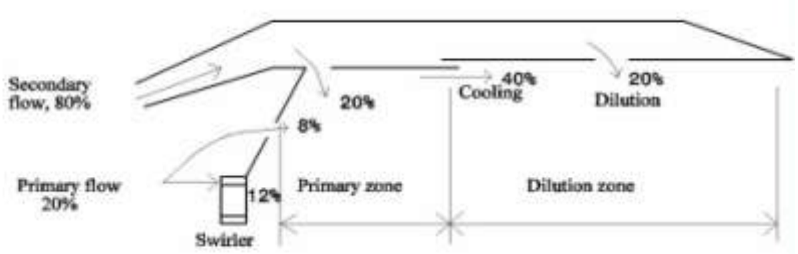

Fig 4.0 Apportioning of combustion air

The basic design details of the combustion chamber is provided in Table 1.0.

Table 1.0 Combustor basic design details

Combustor type	Annular
Combustor inlet air temperature	562 K
Combustor exit temperature	1250 K
Air mass flow	6.0 kg/s
Compression ratio	8.4
Assumed pressure loss	6%

IV. GOVERNING EQUATIONS

The flow is governed by the elliptic partial differential equations. The governing equations represent mathematical statements of conservation laws of physics. The fluid is treated as continuum. The conservation forms of RANS equations for incompressible turbulent flow are;

Continuity : div U = 0

x- momentum : $$\frac{\partial U}{\partial t} + div(UZ) + div\left(\overline{u'b'}\right) = -\frac{1}{\rho}\frac{\partial P}{\partial x} + vdiv(grad(U))$$

y- momentum : $$\frac{\partial V}{\partial t} + div(VZ) + div\left(\overline{v'b'}\right) = -\frac{1}{\rho}\frac{\partial P}{\partial y} + vdiv(grad(V))$$

z- momentum : $$\frac{\partial W}{\partial t} + div(WZ) + div\left(\overline{w'b'}\right) = -\frac{1}{\rho}\frac{\partial P}{\partial z} + vdiv(grad(W))$$

Where, U, V and W are velocity components in Cartesian coordinate systems. Z is mean velocity component. The over bared quantity is fluctuating velocity components introduced due to time averaging and the term $div\left(\overline{v'b'}\right)$ is called Reynolds stresses[11].

V. TURBULENCE MODELING

The standard k-Ɛ two equation model proposed by Launder and Spalding is used for modeling turbulence[12].The robustness and reasonable accuracy with minimum computing resources and economy make this model widely accepted in industrial applications. The model assumes the flow is fully turbulent and the effect of molecular viscosity is negligible. The turbulent kinetic energy k and the dissipation of kinetic energy Ɛare obtained from two transport equations [13].

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) + C_{2\varepsilon\rho}\frac{\varepsilon^2}{k}$$

Where, $G_k$is the generation of turbulent kinetic energy and $G_b$is generation of kinetic energy due to buoyancy, 'C'a constant.□$_k$ and □$_Ɛ$ are turbulent Prandtl numbers for kinetic energy and dissipation.

VI. COMBUSTOR MODEL AND DISCRETIZATION

The 22.5 deg. azimuthal model is considered for flow analysis. The combustor has 16 injectors placed radially in the doom region. The geometry of the injector, igniter and other structural supports are not included in the computational domain in order to make the model simple. Since the geometry is very complex, the structured meshing is not considered. The domain is discretized with unstructured three dimensional tetrahedral cells. Then the cells are converted to polyhedral cells. The tetrahedral cells have only four neighbors. Hence the computation of gradients at cell center using linear shape function is inaccurate. The main advantage of polyhedral cells is that it has many neighbors, normally in the order of 10 and hence the gradients are better approximated. The discretized model of the combustor without dilution tubes is shown in Fig 5.0.

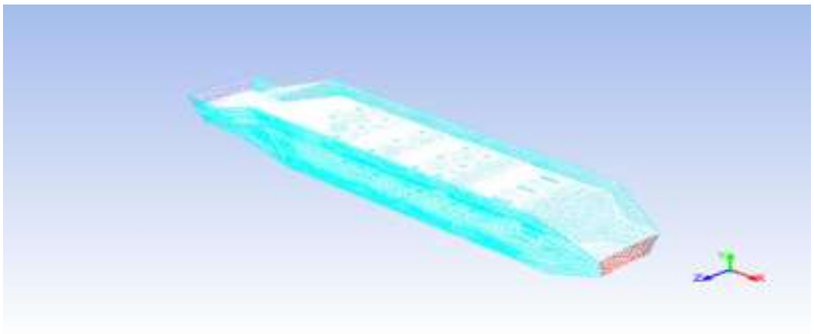

Fig 5.0 Discretized computational domain

The mesh independency study is performed on the combustor without dilution tubes. The three different mesh densities and qualities considered for the study are presented in Table 2.0.

Table 2.0 Mesh quality of combustor without dilution tubes

Case	No. of elements	Maximum edge size	Minimum edge size	Minimum orthogonal quality	Maximum aspect ratio
1	1.00 Million	4.0 mm	0.1 mm	0.17	18.29
2	1.01 M	3.0 mm	0.1 mm	0.16	12.76
3	1.80 M	2.0 mm	0.1 mm	0.10	20.08

The unsteady incompressible Navier-Stokes equations are solved using commercial CFD codes ANSYS to obtain the aerodynamic performance of the combustor. The detailed solution procedure is presented in the next section. The pressure drop determined for the three cases are presented in Table 3.0.

Table 3.0 Combustor pressure loss

Case	Pressure loss	Deviation
1	14.4 %	-
2	14.3 %	0.69 %
3	14.3 %	0.10 %

Since the deviation in the pressure loss is very negligible, the case 3 is considered for further analysis. There are no significant changes in the two combustor geometries which would affect the mesh independency study. Hence no separate mesh independency study is performed on the combustor with dilution tubes. The mesh quality of case 3 is adapted for the combustor with dilution tubes also for detailed analysis.

VII. SOLUTION PROCEDURE

The computational model is applied with cyclic periodic boundary condition to reduce the computing resources and time. The other boundary conditions used in analysis are presented in Table 4.0.The turbulent kinetic energy and turbulent dissipation rate equations are discretized with first orderupwind technique. The pressure based SIMPLE algorithm is used for solving the governing

Table 4.0 Boundary conditions

Boundary	**Type**
Combustor inlet	Inlet mass flow
Combustor exit	Pressure outlet
Casings	Wall with no slip
Working medium	Atmospheric air

equations. This method follows the guess and correct technique to solve the governing equations. The solution procedure is briefly explained.

1. The initial pressure is guessed at each grid point.
2. The momentum equations are solved for velocity components.
3. The pressure correction equation is solved to find corrected pressure.
4. The pressure and velocity are corrected.
5. The previous intermediate values of pressure and velocity are replaced with new corrected values.

This procedure is repeated in an iterative manner till the solution converges on reaching the preset residue, in this case 0.001.

VIII. RESULTS AND DISCUSSION

The flow analysis are performed with the specified air mass flow rate on both the combustor models. The results of air flow analysis performed on the two types of combustor models are discussed in details and compared.The combustor without dilution tubes is referred as Case 1 and the combustor with dilution tubes is referred as Case 2. The X, Y and Z components of velocity across the combustor exit plane along burner axis are presented in Fig 6.0, Fig 7.0and Fig 8.0.

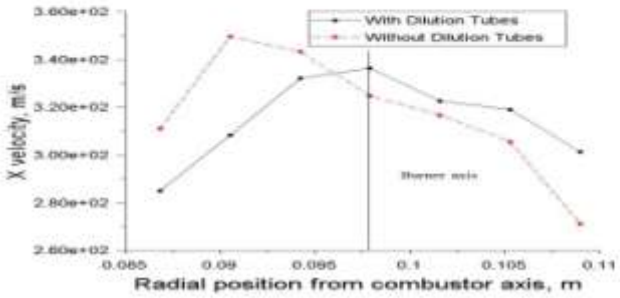

Fig 6.0 X velocity component

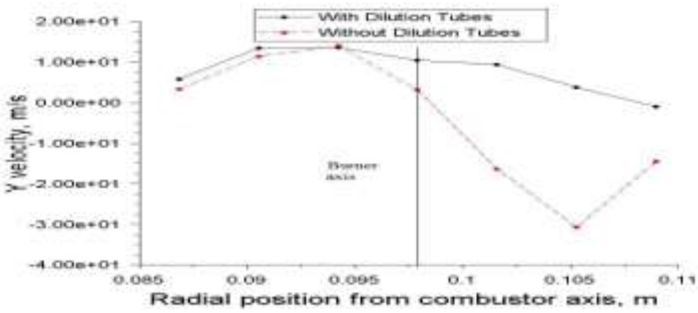

Fig 7.0 Y velocity component

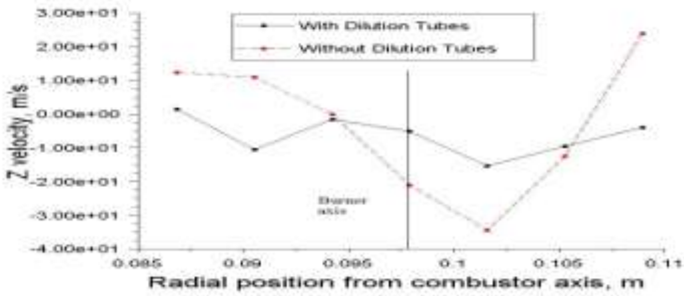

Fig 8.0 Z velocity component

A reasonable similarity is noticed in the X velocity component. The maximum velocity is obtained in case of combustor without dilution tubes and also below the burner axis. Whereas it is maximum near the burner axis in case of combustor with dilution tubes. The Y components of velocity are almost same below the burner axis for both cases. The variation of Y velocity is narrow. The lowest Z velocity is obtained in case of combustor without dilution tubes and noticed above the burner axis.

The surface streamlines are plotted for two combustor models in Fig 9.0 and Fig 10.0.The primary zones in both the models develop reasonably well defined recirculation zones. However the combustor with dilution tubes develops comparatively better recirculation zones.

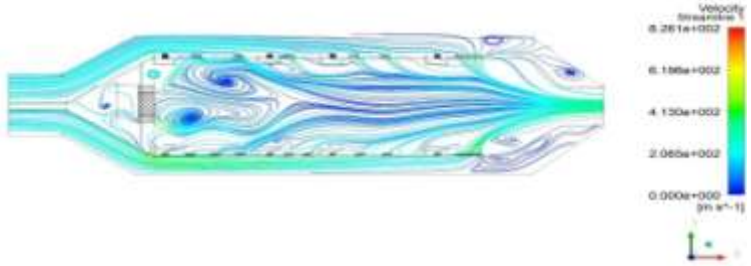

Fig 9.0 Streamlines in combustor, Case 1

Fig 10.0 Streamlines in combustor, Case 2

The recirculation zones play vital role in sustaining the combustion by efficiently mixing the incoming air and in turn enhancing the burning rate.The calculated streamlines are similar to the one reported on can combustor [14].The pressure contour inside the combustor for both the models are almost similar and is shown in Fig 11.0. The total pressure loss across the combustor is found to be 15.0 percent in both the cases.

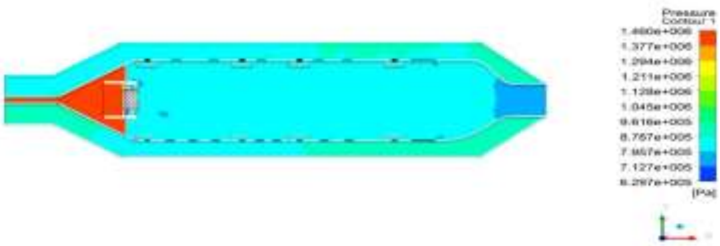

Fig 11.0 Contour plot of pressure

The contour plots of turbulent kinetic energy inside the combustor on meridional plane are shown in Fig 13.0 and Fig 14.0.

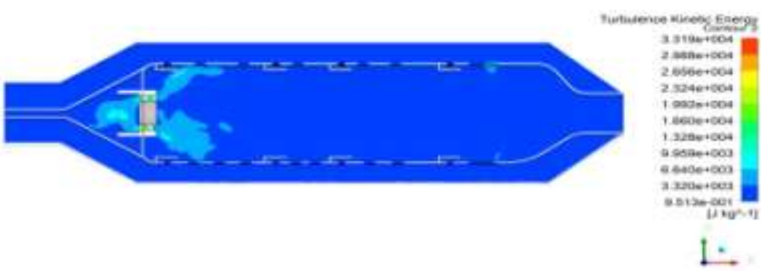

Fig 12.0 Contour of turbulent kinetic energy, case 1

Fig 13.0 Contour of turbulent kinetic energy, case 2

The generation of turbulent kinetic energy is observed in the snout region, recirculation zone and dilution zone. The swirler region experiences more kinetic energy generation. The dissipation of kinetic energy is very less and observed in the swirler region in both combustor models.

IX. CONCLUSION

The two combustor models, one without dilution tubes and the other with dilution tubes are subjected to non-reacting flow analysis. The flow at combustor exit is more uniform in case of combustor with dilution tubes. The recirculation zones are also well structured in the primary zone in both the models. The increase of combustor pressure loss due to introduction of dilution tubes is negligible.Both the models undergo similar level of generation of kinetic energy and it is highly visible in swirler region. The presence of dilution tubes introduces considerable difficulties in combustor manufacturing.Though the acceptable pressure loss in industrial applications is 6.0% to 8.0 %, the pressure loss of 15.0 % calculated in this study is good enough for a preliminary assessment of combustor aerodynamics. The Computational Fluid Dynamics has greatly reduced the time and effort for early assessment of combustor. The fabrication of the combustor by 3D printing and experimental validation of cold flow analysis are envisaged for future research work.

REFERENCES

[1] Lefebvre A.H., Ballal D.R. Gas Turbine Combustion Alternative Fuels and Emissions, CRC Press, 2013.
[2] Shyy W., Braaten M.E., Burrus D.L.Study of three-dimensional gas-turbinecombustor flows, Int. J. Heat Mass Transfer, 32(6):1155-1164, 1989.
[3] Jain A., Choudhary S., Singh S.N., Rai L. Flow analysis in a model can-type gas turbine combustor, Indian Journal of Engineering & Material Sciences, 12: 389-397, 2005.
[4] Melaaen M.C. Three-dimensional numerical analysis of isothermal fluid flow in gas turbine combustors, Int. J. Num. Meth. Heat Fluid Flow, 2: 441-454, 1992.
[5] Lysenko D.A., Solomatnikov A.A. Numerical analysis of aerodynamics and hydraulic loss in the GTE-150 combustion chamber of gas-turbine power plant with the use of the Fluent Suite, Journal of Engineering Physics and Thermophysics, 79(4): 673-677, 2006.
[6] Rodrigues L.O., Alencar H.S., Nascimento M.A.R., Venturini O.J. Aerodynamic analysis using CFD for gas turbine combustion chamber, Proceedings of POWER2007, POWER2007-22181, July 17-19, San Antonio, Texas, 2007.
[7] Gieras M., Stankowski T. Computational study of an aerodynamic flow through a micro-turbine engine combustor, Journal of Power Technologies, 92(2): 68-79, 2012.
[8] McGuirk J.J. The aerodynamic challenges of aeroengine gas-turbine combustion systems, The Aeronautical Journal, 118(1204): 557-599, 2014.
[9] Aligoodarz M., Soleimanitehrani M., Karrabi H., Ehsaniderakhshan F. Numerical simulation of SGT-600 gas turbine combustor, flow characteristics analysis and sensitivity measurement with respect to the main fuel holes diameter, Journal of Aerospace Engineering, 0(0): 1-13, 2016.
[10] Karuppannan S., Mehul B., Sivaramakrishna G., Navindgi R.D., Muthuveerappan N. CFD analysis of flow in a gas turbine combustor swirl cup, Proceedings of the ASME 2017 Gas Turbine India Conference, December 7-8, Bangalore, India, GTINDIA2017-4725, 2017.
[11] Versteeg H., Malalasekra W. An Introduction to Computational Fluid Dynamics, Pearson, India, 2016.
[12] Launder B.E., Spalding D.B. The numerical Computation of Turbulent Flows, Comput. Methods Appl. Mech. Eng., 3: 269-289, 1974.
[13] Ansys Fluent Theory Guide, Release 12.0, pp-47, 2013.
[14] Mare F., Jones W.P., Menzies K.R. Large eddy simulation of a model gas turbine combustor, Combustion and Flame, 137: 278-294, 2004.

Raja Marudhappan1 "No Reaction Flow Analysis Of Aero-Derivative Annular Combustor Of A Turbo Shaft Engine" International Journal of Computational Engineering Research (IJCER), vol. 08, no. 08, 2018, pp. 13-20.