DESIGN AND ANALYSIS OF EXHAUST DIFFUSER OF GAS TURBINE AFTERBURNER USING CFD

[1]VINODKUMAR S HIREMATH, [2]S. GANESAN, [3]R. SURESH

[1]M.Tech. Student, SIT, Tumakuru,
[2]Scientist, GTRE, DRDO, Bangalore,
[3] Associate Professor, Mechanical Engineering Dept., SIT, Tumakuru
E-mail: [1]hiremathvinod99@gmail.com

Abstract— The exhaust diffuser is a part of an afterburner of the gas turbine engine which decreases the velocity of gases coming from the low pressure turbine. The decrease in velocity is required to reduce afterburner cold and hot total pressure loss and to increase flame stability. It helps in better flow control and diffusion in exhaust diffuser leading to increased thrust and combustion efficiency. The present work deals with design and analysis of an exhaust diffuser of gas turbine afterburner. In this study, a baseline diffuser configuration has been designed using 1-D empirical relations. The baseline diffuser configuration has been incorporated into a practical afterburner and the performance of the exhaust diffuser evaluated under non-reacting condition using CFD. Further the diffuser modifications have been carried out using CFD to arrive at a configuration which gives minimum total pressure loss and Mach number. Commercial CFD software, ANSYS FLUENT has been used for the CFD analysis. A significant improvement in performance has been obtained by managing the contours of the diffuser and length. Further a reacting flow analysis has been carried out for the finalized afterburner configuration in order to study the performance of diffuser and the afterburner under reacting (combustion) flow condition. From the analysis, total temperature rise, velocity of flow and thrust of the afterburner has been determined. In addition, the CFD analysis has been extended for different engine bypass ratio and different engine flight conditions of the afterburner to evaluate the diffuser performance.

Index Terms— Afterburner, Exhaust diffuser, CFD, Non-reacting and Reacting flow.

I. INTRODUCTION

An afterburner of a gas turbine engine is a thrust augmenting device used to increase basic thrust of the aircraft engine during supersonic flight or rapid acceleration of the commercial aircraft and for military fighters to improve the combat capability such as steep climbing, sharp turns[1]. The afterburner is located in the downstream part of the engine, after the fan, compressor, combustor and turbine which are main components of the engine. The afterburner increases thrust of the gas turbine engine by adding fuel to the exhaust gases which are having still much un-burnt oxygen. The resultant increase in temperature raises the velocity of gases and hence engine thrust.

A practical afterburner [2] consists of exhaust diffuser, fuel injector, and v-gutter as a flame stabilizer, liner with anti-screech holes and cooling ring holes and nozzle. Gas leaving the turbine is de-swirled and diffused by airfoil struts and diffuser, fuel is added by fuel spray bars (tubes), combustion is initiated in the wake of a number of flame stabilizing devices (flame holders), and the thermal energy of combustion is mixed along flame surfaces spreading outward and downstream from the flame holders. Also, a liner is used in afterburners as both a cooling liner and a screech liner. All engines incorporating an afterburner must also be equipped with a variable area throat exhaust nozzle, in order to provide for proper operation in both afterburning and non-afterburning modes.

II. DESIGN OF DIFFUSER

The diffuser [3] is a device of diverging passage, in which the flow is decelerated and reduction in velocity head (kinetic energy) of flow takes place. The reduction in velocity heads is converted to rise in static pressure. The exhaust diffuser is one of the important component of afterburner, which reduces the velocity of flow to desired value at diffuser exit hence it stabilize the flow. The total pressure loss has a serious impact on engine thrust. Typically, a 1% increase in total pressure loss will result in a 1% decrease in thrust [4]**.**From designers view point an ideal diffuser is one which can achieve the required velocity reduction in shortest possible length, with minimum loss in total pressure and with uniform and stable flow condition at outlet.Large divergence angles can be used because the blockage to flow by the flame holder and fuel injection systems of the afterburner which reduces the tendency of the flow to separate from the diffuser cone.Figure 1 show the geometry and layout of afterburner diffuser configuration considered for the design [5].

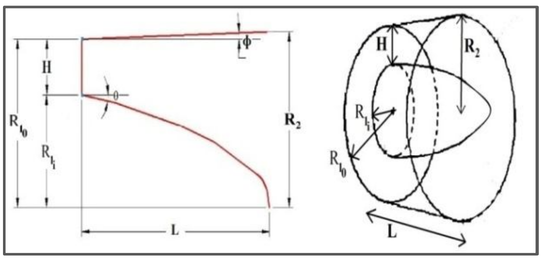

Figure 1 Annular diffuser

The shape of outer radius is defined as

$R_2(x) = R_{1o} + \tan\phi(x)$ (1)

The variation of inner radius of diffuser is defined as, (2)

$r_i^2(x) = tan^2\phi * x^2 + (2\tan\phi * R_{1o} - \frac{2A*tan\theta}{\pi H})\, x + R_{1i}^2$

Where θ=divergence angle is always set equal to sweet spot value [5] and corresponds to θ =4.5°. In designing diffuser, the inlet radius (outer and inner) of diffuser provide by turbine designers. The obstruction (blockages) to the flow is by Struts, Spray bars and Flame holder (V-Gutter). The profile needs to be corrected for blockages in the flow by reducing the radius of the inner profile.

$r_{i,corrected}^2 = \frac{\pi r_i^2 - \text{blockage area}}{\pi}$ (3)

There are number of measures are used to characterize the performance of a diffuser. The pressure recovery coefficient (C_p) and ideal pressure recovery coefficient ($C_{pid)}$ is given by [5]

$C_p = \frac{p_2 - p_1}{p_{t1} - p_1}$ and $C_{pid} = 1 - \frac{1}{AR^2}$

Diffuser efficiency, $\eta_D = \frac{Cp}{Cpid}$

III. AFTERBURNER SYSTEM

Figure 2 shows the model of the afterburner system considered for the current study. The afterburner system consists of core and bypass regions separated by a liner. The salient features of the configuration are:-

- Nine numbers of exhaust cone struts
- Eighteen numbers of spray bars
- V-gutter : nine numbers of radial gutter and one annular gutter
- Nineteen rows of screech holes
- Thirteen rows of cooling rings
- Convergent Divergent (C-D) nozzle

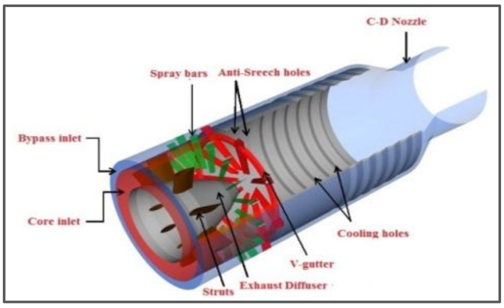

Figure 2 Afterburner model

IV. CFD MODEL DESCRIPTION

The details of the geometric modeling, computational grids and numerical scheme used for the present study are presented in this section.

Geometric Modeling

In this study, a 40° sector model (Figure 3) of the afterburner is considered with appropriate nozzle configuration for non-reacting and reacting conditions. The model includes one strut incorporated with diffuser configuration, two spray bars, and one radial v-gutter attached to annular ring gutter, liner with 19 rows of screech holes each row with 13 holes and 13 rows of cooling rings. In order to apply proper condition at exit, the computational domain is extended downstream of the C-D nozzle.

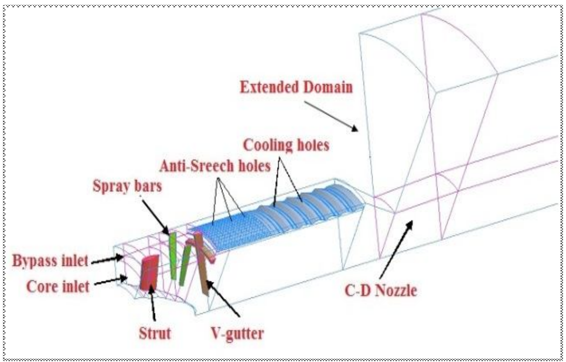

Figure 3 Sector model of the afterburner

Grid

In the present configuration, 3-D, hybrid (combination of structured and unstructured) grids have been generated using GAMBIT software [6]. All the intricate features are meshed as per the geometry viz., strut, spray bar, v-gutter, anti-screech holes, liner cooling holes and nozzle. It has been ensured that the quality of grids in terms of aspect ratio and deviation on orthogonality, is maintained within the required value. Figure 4 shows the grid generated for the present configuration.

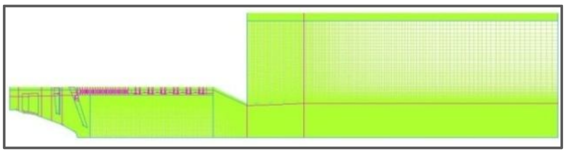

Figure 4 Computational grid

Boundary Conditions

The boundary conditions used for the present analysis are given below:-

Inlet: Mass flow rate is specified at the core inlet and bypass inlet along with total temperature and mass fractions of chemical species are specified.

Outlet: The boundary condition has been applied at the exit of the domain in the form of pressure outlet.

Wall: A wall function approach is used at the wall of exhaust diffuser, v-gutter, spray bars, liner and nozzle. At the walls adiabatic no-slip boundary condition is applied.

Periodic: This boundary condition is applied in pairs to the corresponding faces on the sides of the 0° and 40°.

Turbulence Model: The realizable k-e model has beenused to model the turbulence.

Combustion Model

In the present study, Jet-A ($C_{12}H_{23}$) fuel chemistry is modeled by using a simplified two step chemical reaction scheme. The turbulence-chemistry interaction has been modeled using combined Finite rate and Eddy Dissipation Model. Two step reactions are considered and are as below [7].

$C_{12}H_{23} + 11.75\ O_2 = 12\ CO + 11.5\ H_2O$ (1)

$CO + 0.5\ O_2 = CO_2$ (2)

V. NUMERICAL METHOD

The CFD analysis is carried out using ANSYS software [8]. The Fluent uses a finite volume procedure to solve the Navier-Stokes equation of fluid flow. Flow is modeled to be steady, viscous, compressible and turbulent with and without combustion. The Pressure-velocity coupling has been achieved by SIMPLE algorithm.

VI. RESULTS AND DISCUSSION

In the first part of the study, a non-reacting flow analysis is carried out for the baseline afterburner configuration with designed exhaust diffuser profile. Further the CFD analyses are utilized to modify the profile of exhaust diffuser.

NON-REACTING FLOW ANALYSIS

The afterburner performance is sensitive to Mach number, pressure loss and magnitude of the velocity of the gases flowing around the v-gutter. In the present analysis, the length of the exhaust diffuser and shape is modified with respect to baseline configuration to achieve minimum Mach number and minimum pressure loss. Four diffuser configurations namely D-1 (base-line), D-2, D-3 and D-4 used in the investigation are sketched in Figures 5 and 6.

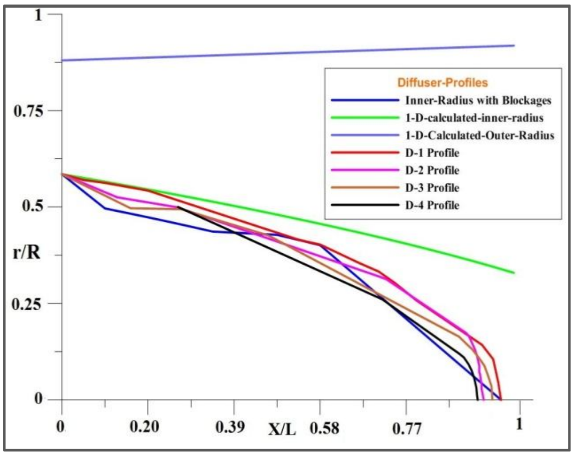

Figure 5 Comparisons of Different Diffuser Profiles

The boundary condition is kept same for all the four cases and the non-reacting flow analysis are repeated. The diffuser performance parameters pressure loss and Mach number at diffuser exit have been evaluated. Using Eq. (1) outer radius profile calculated, Using Eq. (2) Inner radius profile calculated (green line) and Using Eq. (3) inner radius with blockages (dark blue line) calculated. For the base-line (D-1) configuration (red line) the profile has been modified to obtain a smooth variation in area from inlet to exit of the diffuser. All these profiles shown in figure 5

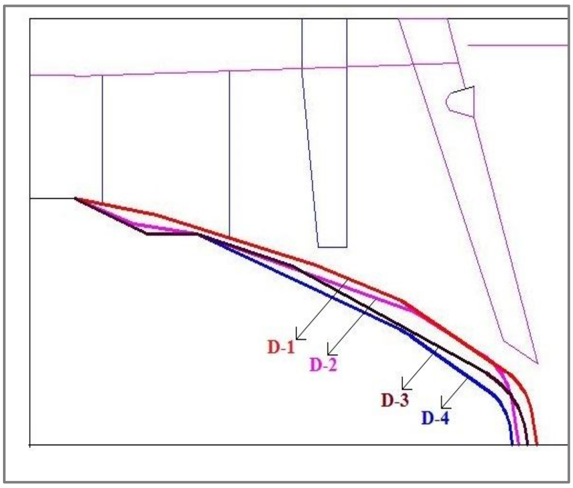

Figure 6 Diffuser profile with afterburner

Figure 7 shows the plot of velocity distribution in the afterburner from inlet to nozzle exit for D-1configuration. It can be seen that the flow decelerates along the diffuser passage. The v-gutter offers obstruction to the flow and result in the formation of low velocity regions behind them. Velocity in combustion zone is relatively low due to the blockage created by the v-gutter. As expected, there is drastic increase in velocity in the Convergent-Divergent (CD) nozzle.

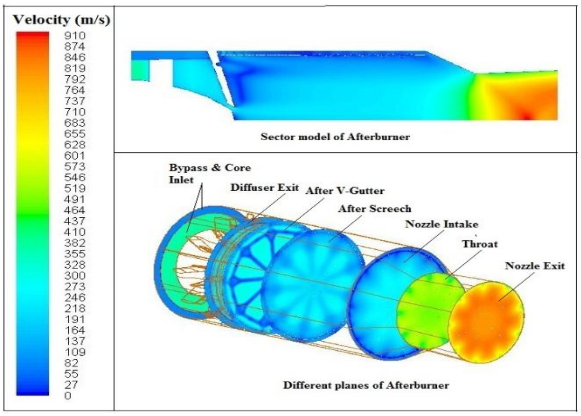

Figure 7 Velocity distribution in the afterburner

Table 1 Velocity, Mach number and pressure loss in the diffuser

Profiles	D-1	D-2	D-3	D-4
Diffuser exit				
Mach Number	0.322	0.320	0.310	0.30
Velocity (m/s)	218	214	208	204
Along Diffuser				
Pressure loss (%)	2.11	2.13	2.31	2.06

The mass weighted average value of inlet velocity and Mach number of flow is 373 m/s and 0.56 respectively. It can be seen from Table 1 that Mach number, velocity of flow and pressure loss is decreases from D-1 to D-4.

Figure 8 shows the static pressure distribution and Figure 9 shows the pressure loss distribution along the afterburner up to V-gutter exit. From Figure 9 lower static pressure recovery is observed from X/L=0 to X/L=1 for D-1 profile. Higher static pressure recovery observed from X/L=0 to X/L=0.23 for D-2 and D-3 profile and Lower static pressure value observed from X/L=0.25 to 1 for D-2 and D-3 profile. A smooth profile with higher static pressure recovery is observed for the D-4 profile from X/L=0.25 to X/L=1. In Figure 9 a similar pressure loss observed from X/L=0 to 0.25 for all diffuser profiles. Compare to other diffuser Lower pressure loss value is observed from X/L=0.25 to 1 for D-4 profile. Compare to both graphs The D-4 profile gives higher static pressure recovery and lower pressure loss in afterburner along diffuser profile.

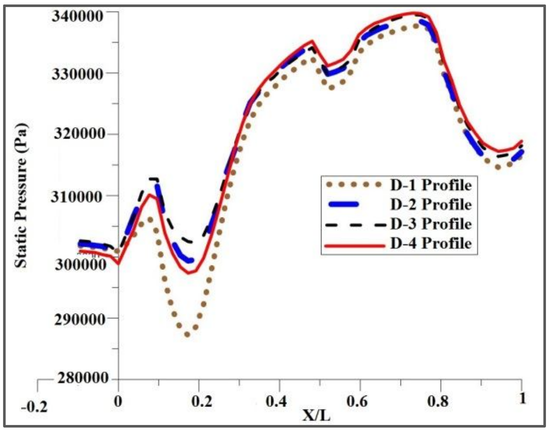

Figure 8 Static pressure (Pa) along axial direction

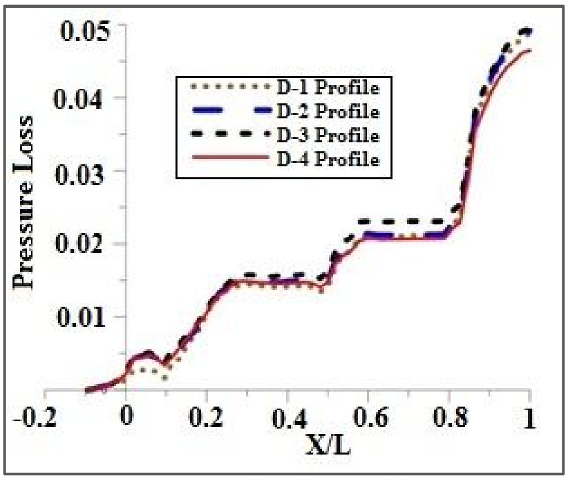

Figure 9 Pressure loss along axial direction

Figure 10shows axial variation of Mach number along axial direction of afterburner up to V-Gutter exit. A higher Mach number is observed from X/L=0 to X/L=1 for D-1 profile. Lower Mach number is observed from X/L=0.05 to X/L=0.23 for D-2 and D-3 profile and higher Mach number is observed from X/L=0.25 to 1 for D-2 and D-3 Profile. A smooth profile with lower Mach number observed for the D-4 profile from X/L=0.25 to X/L=1.

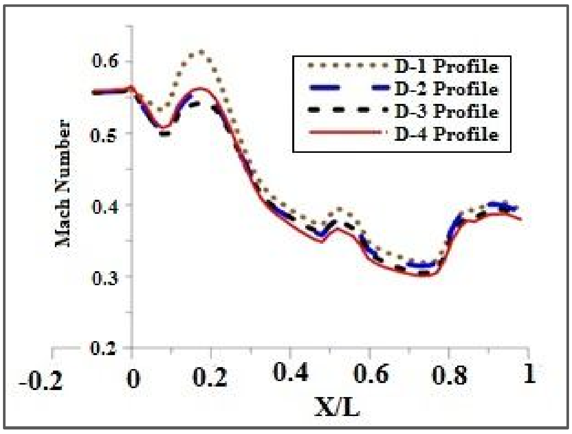

Figure 10 Mach number along axial direction

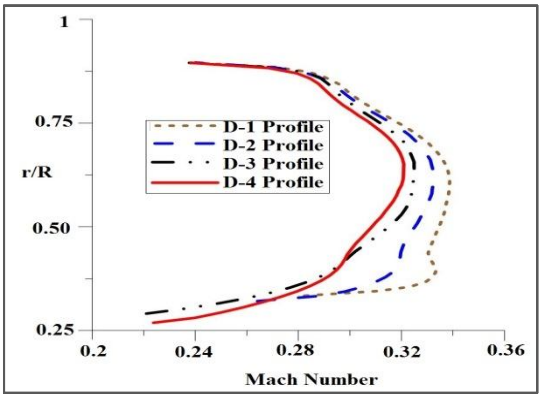

Figure 11 Radial variation of Mach number at diffuser exit

Figure 11 shows radial variation of Mach number at the diffuser exit plane. A higher Mach region is observed from r/R=0.25 to r/R=0.875 for D-1 and D-2 profile and below r/R=0.4 lower Mach is seen for D-3 profile and above r/R=0.4 higher Mach number found. A smooth profile with lower Mach number of flow is observed for the D-4 profile from r/R=0.4 to r/R=0.875. The D-4 profile gives lesser Mach number of flow in afterburner

By observing Table 1 and Figures 8, 9, 10 and 11, the D-4 configuration gives lesser Mach number, velocity and pressure-loss values as compare to other diffuser configurations and henceD-4 diffuser configuration is meeting our design requirements and has been selected for carrying out further analysis.

NON-REACTING FLOW ANALYSIS FOR DIFFERENT BYPASS RATIO:

In this study, the finalized exhaust diffuser profile (D-4) is used to study effect of the bypass ratio on the diffuser performance. Figure 13 shows the radial profile of Mach number at the exit for the different bypass ratios. From figure 12 and Table 2, it can be observed that with change in mass flow rate at inlet the performance of designed exhaust diffuser is unaltered.

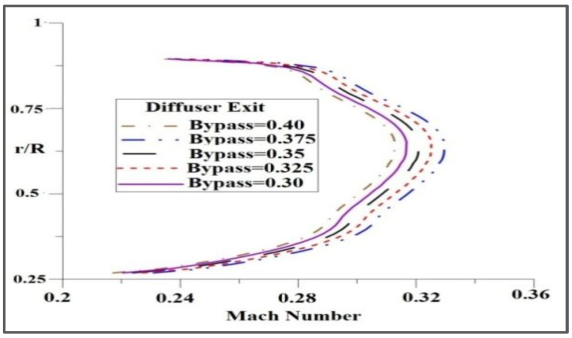

Figure 12 Mach number at diffuser exit

Table 2 Mach number, velocity and pressure Loss at diffuser exit

Bypass Ratio(β)	0.30	0.325	0.35	0.375	0.4
Diffuser Exit					
Mach Number	0.31	0.31	0.30	0.30	0.29
Velocity (m/s)	210	207	204	202	199
Along Diffuser					
Pressure loss (%)	2.25	2.16	2.06	1.99	1.93

NON-REACTING FLOW ANALYSIS FOR FLIGHT CONDITION: In order to study the effect of flight conditions on the diffuser performance, high altitude supercruise conditions is selected. Figure 13 shows the radial profile of Mach number at the diffuser exit. It can be seen that the Mach number variation at diffuser exit is satisfactory. The inlet velocity and Mach number of gases is 365 and 0.53 respectively. At diffuser exit velocity and Mach number of flow reduced to 203 and 0.29. From these, we can concluded that the diffuser configuration arrived is working satisfactorily for different flight conditions.

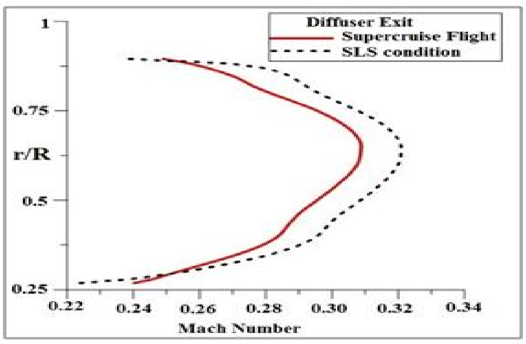

Figure 13 Radial variation of Mach number at diffuser exit

REACTING FLOW ANALYSIS

Reacting flow analysis is carried out for the finalized diffuser (D-4) configuration. In the reacting flow conditions, the nozzle throat and exit area has to be increased so that the afterburner inlet condition and the conditions to upstream components of the afternuburner does not change due to the combustion. Since the core flow is vitiated, the core fluid has been assumed to be composed of O_2, N_2, CO_2 and H_2O only.

Figure 14 shows fuel injection details considered for the present configuration. The fuel is injected from spray bars. Discrete droplet/particle parcels are tracked through the computational domain by solving the Lagrangian conservation equations.

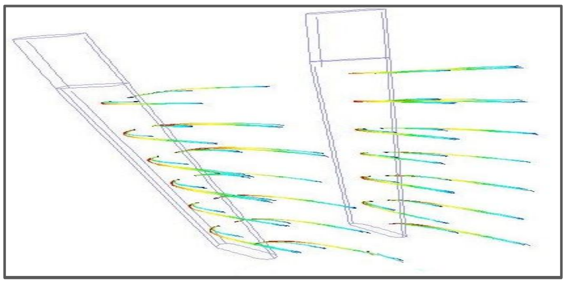

Figure 14 Fuel injection from spray bars

In figure 15 Mach number of reacting flow as compare to non-reacting flow is varying smoothly along radial direction at diffuser exit. In table 3 velocity, Mach number and pressure loss of reacting flow showing similar results as compare to non-reacting flow and from thisit can be concluded that the designed exhaust diffuser performance is unaltered even with fuel injection and reacting flow condition (combustion).

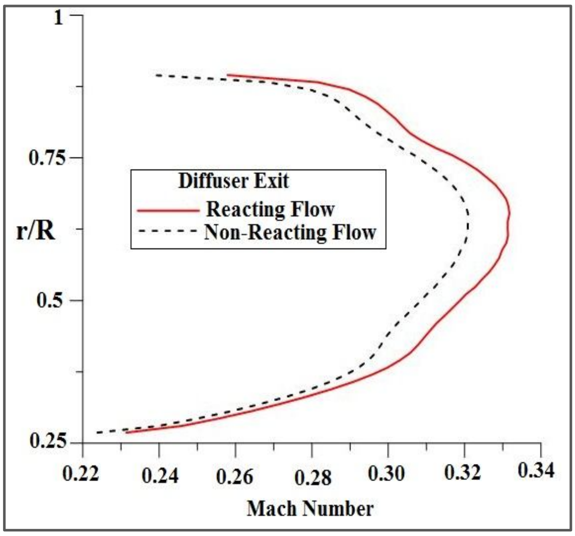

Figure 15 Radial variation of mach number at diffuser exit

Table 3 Mach number and velocity for non-reacting and reacting

Diffuser Exit	Non Reacting-Flow	Reacting Flow
Velocity (m/s)	204	199
Mach Number	0.30	0.31
Along Diffuser		
Pressure Loss (%)	2.06	2.08

Figure 16 shows the velocity magniude in the afterburner for non-reacting and reacting condition. Due to combustion, there is a steep increase in velocity of the flow in the duct from v-gutter to the nozzle exit. Due to increase in the velocity, thrust of afterburner has been improved drastically.

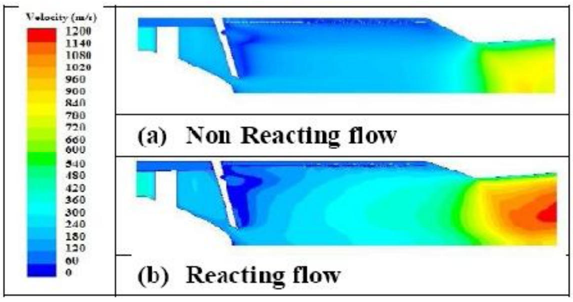

Figure 16 Velocity magnitude in the afterburner for non – reacting and reacting flow

From figure 17 it can be observed that the flame anchors behind the v-gutter and the high temperature region continues downstream direction in reacting flow. The flame behind the v-gutter provides a constant source of ignition and from these sources propagates flame into the fuel air mixture as it flows downstream. It can be clearly seen that the total temperature spread increases along the length of the afterburner and increases from the downstream of the v-gutter to the nozzle entry. The increase in temperatures throughout the afterburner increases the velocity of the gases considerably to attain the required thrust boost.

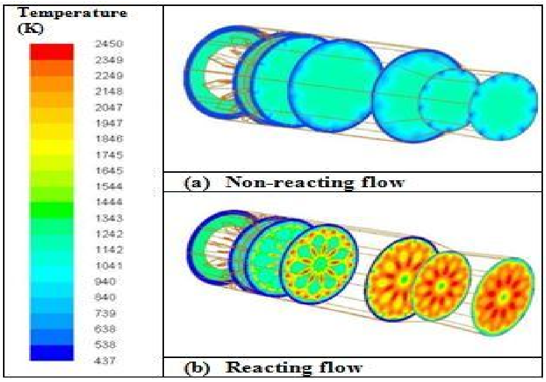

Figure 17 Total temperatures in the afterburner

CONCLUSTION

Numerical predictions have been carried out in the afterburner system under non-reacting and reacting conditions. A baseline diffuser configuration has been designed using empirical relation and incorporated into the CFD model. The diffuser profile modifications have been carried out using CFD and arrived at a suitable configuration for the afterburner system. The diffuser performance checked at different conditions and conformed that it is working satisfactorily at different conditions of afterburner.

NOMENCLAURE

R_{1i} = Inner radius of inlet to diffuser
R_{10} =Outer radius of inlet to diffuser
R_2 =Outer radius of diffuser
Φ =Inclination angle of outer radius
A $= \pi(R_{10}^2 - R_{1i}^2)$
P_1 & p_2 = Static pressure at inlet and outlet
AR = Area ratio= A_2/A_1
p_{t1} =Total Pressure at inlet

REFERENCES

[1] Mattingly J.D., Elements of Gas Turbine, 1996
[2] Housang. B. Ebrahami., "Overview of Gas Turbine Augmentor Design, Operation, and Combustion Oscillation".AIIA,2006
[3] Aurthur H. Lefebvre,Gasturbine Combustion, 2nd edition
[4] J.J. Isaac , N.R. Ramesh , V.S. Krishnakumar , C. Rajashekar , S.R.Shyamsundar , A.P. Haran and V.Sundararajan, A Methodology for Afterburner Evaluation, 3rd National Conference on Air Breathing Engines and Aerospace Propulsion, IIT, Madras,28-30thDec.1996
[5] Jack D. Mattingly, William H. Heiser, David T. Pratt, "Aircraft Engine Design", Second Edition
[6] Gambit User's Guide. Version 2.4.6, 2008
[7] Ganesan. S, Yogesh T.V, Kishore Kumar. S., "Effect of Exhaust Diffuser on Gas Turbine Afterburner Performance", International Conference on Fluid Mechanics and Fluid power, 2010
[8] Ansys Fluent 14.0 Theory Guide and User'sGuide (2011)

★★★