CFD Analysis of Flow in After Burner

DR.N.MOHMED SHERIFF¹, P.SELVA KUMAR², A.H.SYED SULTHAN ALAUEEN³

1. Principal, 2. Senior Lecturer, 3. Student, Department of Mechanical Engineering
Syed Ammal Engineering College
Ramanathapuram, Tamil Nadu -623502.
INDIA.
nmdshariff19@yahoo.com, rpselvakumar@rediffmail.com, www.syedengg.ac.in

Abstract: - After burners are used to accelerate the aircraft during the take off or transonic flight when a temporary increase in thrust required. It is also used to increase the combat capabilities of fighter air craft. The presence of Diffuser struts, Fuel injection rings, Flame holders makes the flow in the after burner more complex. The detailed study of over flow is required for the design and performance evaluation of the after burner. In this work the cold flow analysis of a practical after burner system has been done and the flow around the different parts of the afterburner has been studied. The flow analysis was carried out using compressible, viscous and turbulence mode of commercial CFD software Star-CD. The turbulence is modeled using standard K-ε model. The numerical results were compared with experimental data taken at the after burner front and exit planes. The numerical results agree satisfactorily with the experimental values. This work will be helpful for the Design and Performance evaluation of after burner.

Key-Word: - CFD, Flow Field, Mach number, Nozzle, Swirl, Total Pressure.

1 INTRODUCTION
The three dimensional cold flow analysis has been done for the practical after burner. Hot vitiated gas coming out from the turbine enters the after burner. The gases are diffused in the diffuser to slow down the velocity. After reducing the velocity the fuel is injected by fuel injection rings and burnt, flame stabilization is done by a set of radial and ring gutters. Recirculation of flow is required behind the flame holders to get the stabilized and complete combustion. The gases are sent through the convergence, divergence nozzles to increase the velocity after increasing the total temperature of the after burner.

Major components of after burner namely diffuser, fuel injection rings, flame holders contribute substantial blockage to the flow and hence cause for total pressure loss during dry and wet operation. The gases coming out from the turbine are always having swirl. Since the swirl in the flow cause for the raise in total pressure and reduced performance of the afterburner. Deswirling of gases is essential. Twisted struts used in the after burner geometry are used to deswirl the flow. Hence the detailed study of the three dimensional turbulent flow on the diffuser surface, behind the flame holders is required for the design and development and performance evaluation of the after burner in jet engines.
2 Description of the geometry

Figure 1 is a schematic of the afterburner considered for analysis. Hot vitiated core gases from low pressure turbine enter the annulus of the exhaust diffuser having nine twisted struts at the inlet and four manifolds and complex assembly of ‘V’ Gutters having twelve outer arms, six inner arms and one annulus ring. Fig.2 shows the v-gutter geometry. Cold air enters through the bypass duct and mixes with the hot core flow through chute, screech, and cooling holes located along the liner that separates the core and bypass streams. The remaining bypass flow enters the core region at the CD nozzle entry and leaves the nozzle with high velocity. Also there are five stiffeners in the liner.

3 Description of the CFD Analysis

3.1 Sector Model

Due to symmetry in the geometry, it suffices to model a 120° sector, which includes four outer V-gutters and two inner V-gutters. The twisted struts of two half struts and two full struts are included in the present 120° sector. In order to apply proper condition at exit, the computational domain is extended downstream of the nozzle to a distance of three times of the nozzle exit diameter in the axial direction and two times in the radial direction.

3.2 Grid Generation

The grid for the present geometry is generated using meshing features of STAR-CD software [2]. Three dimensional body-fitted and structured grids of one million mesh points are used for the present analysis. The numbers of grids are selected based on the earlier experience of the analysis of a 60 degree sector afterburner where grid independence studies and comparison with experiments was carried out. Fig.3 shows the meridional view of the grid and Fig.4 shows the full 3-D view of the grid. Fig.5 shows the struts model in the exhaust diffuser and Fig.6 shows the v-gutter. Fig.7 shows the surface grid near exhaust diffuser and v-gutter. Fig.8 shows the meridional view of the struts and v-gutter. Since the liner holes are very small compared to the overall geometry of the after-burner, porous medium approach [1] is used to model the screech holes and cooling rings.
Fig. 4 Three dimensional view of the full grid

Fig. 5 Struts surface grid

Fig. 6 Struts surface grid

Fig. 7 Surface grid struts, v-gutter and bypass

Fig. 8 Meridional view of the v-gutters and struts

Fig. 9 Grid near chute

Fig. 10 Grid near cooling rings
3.3 Boundary conditions

3.3.1 Inlet

At the inlet total pressure, total temperature and swirl angle variation are specified both core and bypass regions. The inlet conditions used for the core are given in Table-1. At the core region, the profile total pressure, temperature swirl angle variations are specified in the radial direction and are assume uniform in the circumferential direction. Uniform values are provided in the radial and circumferential direction for the bypass region. Table-2 shows conditions used for the bypass region.

<table>
<thead>
<tr>
<th>Radius (m)</th>
<th>Total Pressure (bar)</th>
<th>Total Temperature (K)</th>
<th>Swirl Angle (deg.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2295</td>
<td>1.83</td>
<td>820.7</td>
<td>-20.67</td>
</tr>
<tr>
<td>0.243</td>
<td>1.999</td>
<td>860.9</td>
<td>-20.67</td>
</tr>
<tr>
<td>0.2695</td>
<td>2.162</td>
<td>892.7</td>
<td>-20.39</td>
</tr>
<tr>
<td>0.296</td>
<td>2.257</td>
<td>938.7</td>
<td>-18.47</td>
</tr>
<tr>
<td>0.3225</td>
<td>2.280</td>
<td>953.2</td>
<td>-13.25</td>
</tr>
<tr>
<td>0.349</td>
<td>2.272</td>
<td>925.7</td>
<td>-4.21</td>
</tr>
<tr>
<td>0.356</td>
<td>2.287</td>
<td>910</td>
<td>-2.1</td>
</tr>
</tbody>
</table>

Table - 1 : Core region

<table>
<thead>
<tr>
<th>Radius (m)</th>
<th>Total Pressure (bar)</th>
<th>Total Temperature (K)</th>
<th>Swirl Angle (deg.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.356-0.405</td>
<td>2.45</td>
<td>457.5</td>
<td>0</td>
</tr>
</tbody>
</table>

Table - 2 : Bypass region

3.3.2 Outlet

The ambient pressure of 0.91009 bars is imposed at the outlet.

3.3.3 Solid Wall

No slip condition is used on the entire solid wall.

3.3.4 Symmetry plane

The axis of the $120^\circ$-sector model was assigned symmetry condition.

3.3.5 Cyclic

Cyclic boundary conditions are applied at 0 degree and 120 degree sector.

3.4 CFD Code

STAR-CD software is used to solve the 3D NS equations along with the boundary conditions equations. For the present analysis, k-$\varepsilon$ turbulence model with wall function approach is used to simulate turbulence and a second order accurate MARS scheme is used.

3.5 Computational Platform and convergence

The computations have been carried out on a parallel cluster machines (Pentium-IV) (2 GB RAM) with four processors. Convergence has been achieved in about 10930 iterations, which took about 190 hours for the analysis of the present configuration.

3.6. Governing Equations

The flow in afterburner was assumed as steady, 3-D and turbulent. The flow is governed by the conservation equations of mass, momentum and energy, turbulent kinetic energy and its dissipation rate. The general form of these conservation equations can be written as,

$$\nabla (\rho V \phi) - \Gamma_\phi \ \nabla \phi = S_\phi$$
Where
\[ \phi = \text{Any conserved variable} \]
\[ \rho = \text{Density} \]
\[ \vec{V} = \text{Velocity vector} \]
\[ \Gamma_\phi = \text{Exchange coefficient for } \phi \]
\[ S_\phi = \text{Source term coefficient} \]

Various equations can be modeled by suitably formulating the diffusion coefficient \( \Gamma_\phi \) and the source term \( S_\phi \). The dependent variable \( \phi \) stands for velocity components \( U, V, W \), enthalpy \( H \), turbulent kinetic energy \( K \) and dissipation rate \( \varepsilon \).

The turbulence was modeled by a standard \( k-\varepsilon \) turbulence model in which the turbulent viscosity is calculated as

\[ \mu_t = f_\mu \frac{c_\mu \rho k^2}{\varepsilon} \]

The conservation equations were solved by the code STAR-CD, which employs finite volume method and a simple algorithm governing equation.

**4 EXPERIMENTAL RESULTS**

The engine is instrumented at the required locations in the engine. The instrumentation scheme of the Jet pipe and the nozzle is shown in the Fig 12. The Jet Pipe Front plane is instrumented with 3 total pressures and 4 total temperature rakes with 8 points each. It has 2 and 5 wall static pressure orifices on the jet pipe casing (the bypass flow) and the liner (forming core flow) respectively (Fig 12). The Jet Pipe Exit plane (nozzle inlet plane) is instrumented with 3 total pressures and 4 total temperature rakes having 9 and 10 points each. It has 2 and 3 wall static pressure orifices on the liner and the jet pipe casing respectively (Fig 13). There are 9 wall static pressure orifices on the CD nozzle in the after burner as shown in the Fig 14.
5. RESULTS AND DISCUSSION

In the following, results obtained from CFD analysis in various zones are depicted and discussed. Fig. 15 shows the circumferential view with both struts and v-gutter and various locations where the flow is depicted in the following figures. The velocity vector plot in an axial plane ($\theta = 0^\circ$) is shown in figure 16 from jet pipe entry to exit of the extended domain.

The flow diffuses from the inlet to exit of the diffuser and accelerates in the CD nozzle.

Fig. 17 shows the velocity vectors in the CD nozzle and extended domain. It can be observed that the magnitude of velocity is low in the extended domain indicating the adequacy of the extended domain in the vertical direction. Similar behavior in the flow is seen in other planes in the circumferential direction.

The flow in the CD nozzle is shown in fig.18 where the increase in velocity is seen from the inlet of CD nozzle to the exit plane. The Mach number distribution in the CD nozzle in three planes is shown in fig 19 to fig 21. In these figures the acceleration of the flow from sonic at the throat to supersonic in the divergent portion can be observed.

The swirl distribution at the inlet of the jet pipe, inlet and exit of the CD nozzle is shown in fig 22, 23 and 24. The swirl reduces substantially from the inlet distribution of ($-21^\circ$, $0^\circ$) to ($-10^\circ$, $0^\circ$) at the CD nozzle inlet. It further reduces to ($-3.5^\circ$, $0^\circ$) at the CD nozzle exit.
Fig. 16 Velocity vectors at 0° plane from inlet to exit of the extended domain

Fig. 17 Velocity vectors at 0° plane in the CD nozzle and the extended domain

Fig. 18 Velocity vectors at 0° plane in the CD nozzle

Fig. 19 Mach number contours at 0° plane in the CD nozzle

Fig. 20 Mach number contours at 30° plane in the CD nozzle

Fig. 21 Mach number contours at 60°
6 Experimental Validations

Measurements of total pressure and total temperature are taken at the jet pipe front plane and jet pipe exit plane shown in Fig 11. In figure 25 to fig 28, the experimental and the CFD results of the total pressures and total temperatures are shown. It is observed that there is a close match between the experiments and CFD. The static pressure distribution of the CFD results along with the measurements on the CD nozzle wall is shown in figure 29. A close match is seen up to the throat after which a deviation can be seen between the CFD results and the experiments.
Fig. 25 Total pressure variation at the jet pipe front plane

Fig. 26 Total temperature variation at the jet pipe front plane

Fig. 27 Total pressure variation at the jet pipe exit plane

Fig. 28 Total temperature variation at the jet pipe exit plane
7 Conclusion

The three dimensional cold flow analysis has been successfully done for a practical afterburner system. The flow field shows that desired wakes are formed behind the flame holders. The velocity increases in the CD nozzle along its length and reaches its maximum value at the exit plane of the nozzle. The flow is highly deswirled at the jet plane exit nozzle. The computed total pressure, total temperature at the afterburner front and exit planes, wall static pressure in the CD-nozzle has been compared with the experimental results. The numerical results quit satisfactorily agree with the experimental values.

References:


