RESEARCH ARTICLE

Numerical Analysis of Subsonic Coaxial Jet on Effect of Potential Core Length

*G Saravanan¹, A Vinoth Kumar¹, K.B Ravichandrarumar¹
¹Faculty, Department of Aerospace Engineering, SRM University, Chennai, India.

Received-18 January 2016, Revised-17 February 2016, Accepted-26 February 2016, Published-26 February 2016

ABSTRACT

The aim of this project is to study the effect of flow inside the subsonic co-axial nozzle used in aircraft turbofan engine. The study has been carried out by varying the Mach number of secondary flow by maintaining the constant Mach number of primary flow. The coaxial nozzle of the circular section was designed and analysed using CATIA and ANSYS V15.0 software for a Mach number range of 0.3 to 0.6 with secondary sub-sonic flow. The pressure and velocity contours were investigated along the flow. The increase in the velocity difference between the primary and secondary flow results in more turbulence which in turn decreases the potential core length.

Keywords: Coaxial, Potential core, Turbulence, Mach number, Nozzles.

1. INTRODUCTION

Coaxial jets are simple in configuration which accelerates subsonic fluids. [1] Coaxial jets are widely used to provide mixing between fuel and oxidisers in the combustor of aerospace propulsion system and gas turbine engines. The effective mixing of air and fuel will provide the best overall combustion parameters. [2] In a gas turbine engine nozzle, the nozzle pressure ratio is high enough such that the flow will have sonic velocity at throat and reaches up to Mach 1 and the nozzle is said to be choked. Further increase in pressure ratio will increase the Mach number above 1. Compared with single jets, coaxial jets with round nozzles can develop flow structures of very different topology.

[3] The coaxial flow behaviour of fluids is of major concern in many engineering applications. This enhances the mixing flow issuing from the exhaust with ambient air.

[4] In dual stream jets a high speed primary flow is surrounded by a secondary flow. [5] In a coaxial jet, mixing is achieved mainly due to the velocity ratio, density ratio, compressibility and turbulence levels of the two streams, swirl, pressure gradient and free shear flows. The potential core length (length up to which the effect of shock waves exist from the nozzle exit) will be more.

Coaxial jets are more effective in producing turbulence. Entraining of jet flow with atmospheric air is improved by increasing the turbulence. They reduce noise by providing shielding effect to potential flow. They are also used to increase the thrust by reducing potential core length of primary flow [6, 7, 8].

CFD analysis has become more popular now a days to analyse physical phenomenon and to solve associated engineering problems. This increasing use of CFD techniques in turbulence measurements requires high quality experimental data and using those validations of particular model can be incorporated [9]. [10] used pitot probes to obtain experimental data within the jet plume and surrounding flow field. [11] used numerical approach to create a methodology for the direct calculation of noise from realistic nozzle geometries. [12] used the concept of numerical study in the tube developing region. Against the background of studies by various groups conducted on mixing enhancement of turbulent jets, the present paper aims to
contribute a definitive experimental study which compares the respective mixing performances of circular convergent nozzles having different velocity ratios. A CFD model using fluent commercial code linear k-ε standard model and standard wall functions was developed. The results obtained by CFD are used to find the potential core length results. This study also aims to ascertain the ability of the CFD model to adequately predict the mixing characteristics such as potential core length, velocity profile, radial and centre-line pressure and jet half width under varying velocity ratios.

2. METHODOLOGY

2.1. Geometry

The coaxial nozzle shapes were designed using ANSYS Design Modeller 15.0. The coaxial nozzle arrangements employed a fixed inner (primary) nozzle and outer (secondary) nozzle. The primary nozzle has an exit diameter of 1 mm and is designed using Area-Mach number relation. The domain length and diameter arrangement is 300 mm and 200 mm respectively. Figure 1 shows the coaxial nozzle geometry.

![Coaxial nozzle geometry](image)

Figure 1. Coaxial nozzle geometry

2.2. Computational grids

The solution-adaptive mesh refinement feature of Fluent allows us to refine and coarsen our grid based on geometric and numerical solution data. In addition, Fluent provides tools for creating and viewing adaption fields customized to particular applications. By using solution-adaptive refinement, the grid independency study has been carried out.

2.3. Computation method

The 2-D analysis was carried out with the density based solver for compressible jet flows. The linear k-ε standard model and standard wall functions are employed to calculate the viscous effect in the jet flows.

The SIMPLE (Semi Implicit Method for Pressure Linked Equation) shown in equation in (2.1) were utilised for analysis.

\[
\frac{\partial (ur)}{\partial x} + \frac{\partial (vr)}{\partial r} = 0;
\]

\[
u \frac{\partial (u)}{\partial x} + v \frac{\partial (u)}{\partial r} = \frac{1}{r} \nu_t r \frac{\partial u}{\partial r} \quad (2.1)
\]

where \( u \) and \( v \) are the velocities in the axial and radial directions and \( \nu_t \) is eddy viscosity. The Fluent package was used for CFD analysis.

2.4. Boundary conditions

The inflow boundary condition of nozzle has been taken as the pressure inlet and outflow condition has been taken as the pressure outlet with no slip wall boundary condition. Table 1 shows the boundary conditions. Figure A1 shows the schematic of boundary conditions. The pressure conditions for secondary flow were given by the formula shown in (2.2),

\[
P_s = \left[ 1 + \frac{\gamma - 1}{2} M_s^2 \right]^{\frac{\gamma}{\gamma - 1}} \quad (2.2)
\]

<table>
<thead>
<tr>
<th>Secondary flow Mach Number</th>
<th>Primary Flow Velocity (m/s)</th>
<th>Secondary flow Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.6</td>
<td>208</td>
<td>208</td>
</tr>
<tr>
<td>0.51</td>
<td>208</td>
<td>177</td>
</tr>
<tr>
<td>0.45</td>
<td>208</td>
<td>156.2</td>
</tr>
<tr>
<td>0.39</td>
<td>208</td>
<td>135.36</td>
</tr>
<tr>
<td>0.36</td>
<td>208</td>
<td>124.95</td>
</tr>
<tr>
<td>0.3</td>
<td>208</td>
<td>104.14</td>
</tr>
</tbody>
</table>

3. RESULTS AND DISCUSSION

The central core of the jet where the exit velocity is preserved up to some axial distance is called as potential core length. The formation of potential core length is visualised in the velocity plot as shown in figures A2-A7.

The analysis was carried out for various Mach number till the solutions get converged. The model analyzed were used to study the effect of potential core length by varying the secondary flow Mach number. In all the cases, the potential core of the primary nozzle results in decrement with increase in the velocity of the secondary flow.
The velocity contours of secondary flow by varying the Mach number between 0.3 and 0.6 are shown in the figures A2-A7.

The velocity contour for hot flow of Mach number 0.6 and cold flow in outer nozzle Mach number of 0.6 is shown in figure A2. In this the potential core is clearly visible in red colour which represents the potential core region.

The velocity contour for hot flow of Mach number 0.51 and cold flow in outer nozzle Mach number of 0.6 is shown in figure A3. In this the potential core is clearly visible in red colour which represents the potential core region decrease as compared to figure A2.

The velocity contour for hot flow of Mach number 0.45 and cold flow in outer nozzle Mach number of 0.6 is shown in figure A4. In this the potential core is clearly visible in red colour which represents the potential core region decrease as compared to figure A3.

The velocity contour for hot flow of Mach number 0.39 and cold flow in outer nozzle Mach number of 0.6 is shown in figure A5. In this the potential core is clearly visible in red colour which represents the potential core region decrease as compared to figure A4.

The velocity contour for hot flow of Mach number 0.36 and cold flow in outer nozzle Mach number of 0.6 is shown in figure A6. In this the potential core is clearly visible in red colour which represents the potential core region decrease as compared to figure A5 but the rate of decrement is less compared to the other ratio.

The velocity contour for hot flow of Mach number 0.31 and cold flow in outer nozzle Mach number of 0.6 is shown in figure A7. In this the potential core is clearly visible in red colour, but there is no change in the potential core length after the velocity ratio.

It is observed that the velocity difference between the primary and secondary flow affects the length of the potential core due to increase in the turbulent viscosity. From this, primary jet velocity decay rate can be understood. The potential core length of the flow will be more, if jet decays at a slower rate and vice versa.

The decay of mean velocity along the jet centerline at different stations downstream from X/D=0 to X/D=125 for outer nozzle Mach number of 0.51 is shown in figure 3.

The velocity contour for hot flow of Mach number 0.45 and 0.39 is shown in figures 4 and 5, which is almost same because there is no change in the potential core length for the velocity ratio greater than this for the mentioned temperature.

The decay of mean velocity along the jet centerline at different stations downstream from X/D=0 to X/D=125 for outer nozzle Mach number of 0.6 is shown in figure 2.
From the figures 2-5 we observe that the potential core length is decreased up to 25 percent for the particular velocity ratio is comparatively higher.

4. CONCLUSION

The conclusion of this study indicates that analysis employing linear two-equations turbulence modelling can predict the effect of spreading rates of low-speed coaxial jets reasonably well. The knowledge gained in the computational approach enabled the examination of turbulent kinetic energy in the developing jet.

It was observed that decreasing the inlet–cold pressure ratio and keeping inlet-hot at a constant pressure gives a high velocity at the exit due to decrease in pressure at the inlet of the nozzle. Finally, we can conclude that increasing the pressure ratio of secondary to primary nozzle increases the spreading rate. This results in less turbulence which in turn results in larger potential core length.

REFERENCES


APPENDIX A

Figure A1. Boundary conditions

Figure A2. Velocity contours for hot flow 0.6Mach and cold flow 0.6Mach
Figure A3. Velocity contours for hot flow 0.6Mach and cold flow 0.51Mach

Figure A4. Velocity contours for hot flow 0.6Mach and cold flow 0.45Mach
Figure A5. Velocity contours for hot flow 0.6Mach and cold flow 0.39Mach

Figure A6. Velocity contours for hot flow 0.6Mach and cold flow 0.36Mach

Figure A7. Velocity contours for hot flow 0.6Mach and cold flow 0.3Mach