THE PREDICTION OF NOISE AND INSTALLATION EFFECTS
OF HIGH-SUBSONIC DUAL-STREAM JETS IN FLIGHT

A Dissertation in
Aerospace Engineering
by
Swati Saxena

© 2012 Swati Saxena

Submitted in Partial Fulfillment
of the Requirements
for the Degree of

Doctor of Philosophy

December 2012
The dissertation of Swati Saxena was reviewed and approved* by the following:

Philip J. Morris  
Boeing/A.D. Welliver Professor of Aerospace Engineering  
Dissertation Advisor, Chair of Committee

Dennis K. McLaughlin  
Professor of Aerospace Engineering

Kenneth S. Brentner  
Professor of Aerospace Engineering

Victor W. Sparrow  
Professor of Acoustics  
Interim Chair of the Graduate Program in Acoustics

George A. Lesieutre  
Professor of Aerospace Engineering  
Head of the Department of Aerospace Engineering

*Signatures are on file in the Graduate School.
Abstract

Both military and civil aircraft in service generate high levels of noise. One of the major contributors to this noise generated from the aircraft is the jet engine exhaust. This makes the study of jet noise and methods to reduce jet noise an active research area with the aim of designing quieter military and commercial aircraft. The current stringent aircraft noise regulations imposed by the Federal Aviation Administration (FAA) and other international agencies, have further raised the need to perform accurate jet noise calculations for more reliable estimation of the jet noise sources. The main aim of the present research is to perform jet noise simulations of single and dual-stream jets with engineering accuracy and assess forward flight effects on the jet noise. Installation effects such as caused by the pylon are also studied using a simplified pylon nozzle configuration.

Due to advances in computational power, it has become possible to perform turbulent flow simulations of high speed jets, which leads to more accurate noise predictions. In the present research, a hybrid unsteady RANS-LES parallel multi-block structured grid solver called EAGLEJet is written to perform the nozzle flow calculations. The far-field noise calculation is performed using solutions to the Ffowcs Williams and Hawkings equation. The present calculations use meshes with 5 to 11 million grid points and require about three weeks of computing time with about 100 processors. A baseline single stream convergent nozzle and a dual-stream coaxial convergent nozzle are used for the flow and noise analysis.

Calculations for the convergent nozzle are performed at a high subsonic jet Mach number of $M_j = 0.9$, which is similar to the operating conditions for commercial aircraft engines. A parallel flow gives the flight effect, which is simulated with a co-flow Mach number, $M_{cf}$ varying from 0.0 to 0.28. The grid resolution effects,
statistical properties of the turbulence and the heated jet effects ($TTR = 2.7$) are studied and related to the noise characteristics of the jet. Both flow and noise predictions show good agreement with PIV and microphone measurements. The potential core lengths and nozzle wall boundary characteristics are studied to understand the differences between the numerical potential core lengths as compared to experiments. The flight velocity exponent, $m$ is calculated from the noise reduction in overall sound pressure levels (OASPL, $dB$) and relative velocity ($V_j - V_{cf}$) at all jet inlet (angular) angles. The variation of the exponent, $m$ at lower ($50^\circ$ to $90^\circ$) and higher aft inlet angles ($120^\circ$ to $150^\circ$) is studied and compared with available measurements. Previous studies have shown a different variation of the exponent with inlet angles while the current numerical data match well with recent experiments conducted on the same nozzle geometry.

Today, turbofans are the most efficient engines in service used in almost all major commercial aircraft. Turbofans have a dual-stream exhaust nozzle with primary and secondary flow whose flow and noise characteristics are different from that of single stream jets. A Boeing-designed coaxial nozzle, with area ratio of $A_s/A_p = 3.0$, is used to study dual-stream jet noise in the present research. In this configuration, the primary nozzle extends beyond the secondary nozzle, which is representative of large turbofan engines in commercial service. The flow calculations are performed at high subsonic Mach numbers in the primary and secondary nozzles ($M_{pj} = 0.85$, $M_{sj} = 0.95$) with heated core flow, $TTR_p = 2.26$ and unheated fan flow, $TTR_s = 1.0$. The co-flow of $M_{cf} = 0.2$ is used. The subscript $p$, $s$ and $amb$ represent the primary (core) nozzle, the secondary (fan) nozzle, and the ambient flow conditions, respectively. The statistical properties in the primary and secondary shear layers are studied and compared with those of the single stream jets. It has been found that the eddy convection velocity is lower in dual-stream jets as compared to the single stream jet operating at a similar jet exit Mach number. The phase velocity is higher in the secondary shear layer as compared to primary shear layer. The noise measurements agree well with the predicted data and noise reduction is observed in the presence of co-flow. The variation of the flight velocity exponent is calculated as a function of nozzle inlet angle. The value of the exponent at higher inlet angles is lower as compared to the single stream jets. This suggests that the noise levels are less affected in the peak noise direction in the presence of co-flow in dual-stream jets as compared to single stream jets. Two reference velocities: primary jet exit velocity $V_{pj}$ and mixed velocity $V_{mix}$ are considered which result in different absolute values of the exponents. Scaling of the jet spectra is performed at different inlet angles and good collapse has been obtained between the spectra.
The installation effects on jet noise are studied using a simplified pylon structure with a dual-stream nozzle. In the presence of a pylon, the azimuthal symmetry of the nozzle is lost and thus the flow characteristics are different as compared to the baseline nozzle. This will result in different noise characteristics of the installed jet.
Table of Contents

List of Figures x
List of Tables xxii
List of Symbols xxiv
Acknowledgments xxviii

Chapter 1

Introduction 1
1.1 Literature Review: Subsonic Jet Noise .......................... 6
  1.1.1 Single-Stream Jets ...................................... 6
  1.1.2 Dual-Stream Jets ....................................... 10
  1.1.3 Dual-Stream Jets with Installation Devices .............. 14
1.2 Noise Calculations ............................................ 18
1.3 Computational Aeroacoustics Issues ............................ 22
1.4 Objectives of the Current Work ................................ 24
1.5 Original Contributions of the Current Work .................. 24
1.6 Outline of the Thesis ......................................... 26

Chapter 2

Numerical Methods 28
2.1 Hybrid RANS/LES Turbulence Modeling ........................ 29
2.2 Dispersion Relation Preserving (DRP) scheme ................ 35
2.3 Time-Marching Schemes ...................................... 38
### Chapter 2

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.3.1</td>
<td>Low-Dissipation Low-Dispersion Runge-Kutta Scheme</td>
<td>38</td>
</tr>
<tr>
<td>2.3.2</td>
<td>Traditional Runge-Kutta Methods</td>
<td>39</td>
</tr>
<tr>
<td>2.3.3</td>
<td>Local Time Stepping</td>
<td>41</td>
</tr>
<tr>
<td>2.3.4</td>
<td>Dual Time Stepping</td>
<td>41</td>
</tr>
<tr>
<td>2.4</td>
<td>Artificial Dissipation</td>
<td>43</td>
</tr>
<tr>
<td>2.4.1</td>
<td>Sixth Order Low Pass Filter</td>
<td>43</td>
</tr>
<tr>
<td>2.4.2</td>
<td>Selective Artificial Dissipation</td>
<td>44</td>
</tr>
<tr>
<td>2.5</td>
<td>Immersed Boundary Method</td>
<td>45</td>
</tr>
<tr>
<td>2.6</td>
<td>Implicit Residual Smoothing</td>
<td>47</td>
</tr>
<tr>
<td>2.7</td>
<td>Multi-Block Topology and Characteristic Interface Boundary Conditions</td>
<td>50</td>
</tr>
<tr>
<td>2.8</td>
<td>Non-Matching Mesh Block Interfaces</td>
<td>54</td>
</tr>
<tr>
<td>2.9</td>
<td>Boundary Conditions</td>
<td>54</td>
</tr>
<tr>
<td>2.10</td>
<td>Flowcs Williams and Hawkins Equation</td>
<td>56</td>
</tr>
<tr>
<td>2.11</td>
<td>EAGLEJet and PSFWH Solvers Development</td>
<td>60</td>
</tr>
<tr>
<td>2.12</td>
<td>Summary</td>
<td>60</td>
</tr>
</tbody>
</table>

### Chapter 3

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Determining Parallel Efficiency of the Solver</td>
<td>63</td>
</tr>
<tr>
<td>3.2</td>
<td>Reflection of a Transient Acoustic Pulse by a Wall</td>
<td>65</td>
</tr>
<tr>
<td>3.3</td>
<td>Supersonic Flow Past a Wedge</td>
<td>68</td>
</tr>
<tr>
<td>3.4</td>
<td>Gaussian Pulse in a Multi-Block Grid</td>
<td>71</td>
</tr>
<tr>
<td>3.5</td>
<td>Turbulent Subsonic Flow Over Flat Plate</td>
<td>74</td>
</tr>
<tr>
<td>3.6</td>
<td>Subsonic Flow Past a Circular Cylinder</td>
<td>76</td>
</tr>
<tr>
<td>3.7</td>
<td>Transonic Flow Over RAE2822 Airfoil</td>
<td>80</td>
</tr>
<tr>
<td>3.8</td>
<td>Periodic Pressure Pulse Extraction on Acoustic Data Surface</td>
<td>83</td>
</tr>
<tr>
<td>3.9</td>
<td>Summary</td>
<td>84</td>
</tr>
</tbody>
</table>

### Chapter 4

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1</td>
<td>Introduction</td>
<td>85</td>
</tr>
<tr>
<td>4.2</td>
<td>Nozzle Case Setup</td>
<td>87</td>
</tr>
<tr>
<td>4.3</td>
<td>Grid Refinement Study</td>
<td>91</td>
</tr>
<tr>
<td>4.4</td>
<td>FW-H Surface Location Study</td>
<td>93</td>
</tr>
<tr>
<td>4.5</td>
<td>Flow Simulation Results</td>
<td>94</td>
</tr>
<tr>
<td>4.6</td>
<td>Statistical Analysis of Turbulence for Jets in Flight</td>
<td>115</td>
</tr>
<tr>
<td>4.7</td>
<td>Noise Predictions and Forward Flight Analysis</td>
<td>123</td>
</tr>
<tr>
<td>4.8</td>
<td>Summary</td>
<td>133</td>
</tr>
</tbody>
</table>
# Chapter 5

**Dual-Stream Jet Noise Simulations**

- 5.1 Introduction .................................. 135
- 5.2 Dual-stream Nozzle Case Setup ............... 137
- 5.3 Flow Simulation Results ........................ 139
- 5.4 Statistical Analysis of Turbulence for Jets in Flight ............. 151
- 5.5 Noise Predictions and Forward Flight Analysis .................. 160
- 5.6 Dual-Stream jet with Installation Effects ............... 168
  - 5.6.1 Case Setup for Nozzle with Pylon ............ 170
  - 5.6.2 Flow Simulation Results ...................... 171
- 5.7 Summary ...................................... 175

# Chapter 6

**Conclusions and Future Work**

- 6.1 Conclusions ................................... 177
  - 6.1.1 Numerical Methods ......................... 178
  - 6.1.2 Jet Noise Simulations ...................... 180
    - 6.1.2.1 Single Stream Jets ..................... 180
    - 6.1.2.2 Dual-Stream Jets and Installation Effects .... 184
- 6.2 Suggestions for Future work .................... 188

# Appendix A

**Governing Equations in General Curvilinear Coordinates**

- 190

# Appendix B

**Boundary Conditions**

- 195
  - B.1 Riemann Invariant Boundary Conditions ............ 195
  - B.2 Radiation Boundary Conditions .................. 198
  - B.3 Methods of Characteristics Boundary Conditions .... 200
    - B.3.1 Fully-matching Block Interface Conditions .... 207
      - B.3.1.1 Same Mesh-Orientation .................. 207
      - B.3.1.2 Arbitrary Mesh-Orientation .............. 209
    - B.3.2 Non-Matching Block Interface Conditions .... 210
  - B.4 Total Boundary Conditions ..................... 213

# Appendix C

**Analytical Solutions**

- 214
  - C.1 Periodic Pressure Pulse Propagation ............ 214
  - C.2 Gaussian Pulse Propagation .................... 215
List of Figures

1.1 (a) Typical subsonic broadband jet noise spectra $SPL$ vs. $St$ for a $M_j = 0.9$ jet obtained at different observer distances by linear propagation. (b) Far-field narrow-band noise spectra of a supersonic jet operating at $M_j = 2.0$. 4

1.2 Visualization of far-field sound: Velocity Gradients in a round subsonic $M = 0.9$ jet calculated using DNS. 7

1.3 (a) Instantaneous contours of vorticity magnitude: levels are $\omega r_o/U_j$ = 0.35, 1, 2, 3, 4 with lighter contours representing larger values. (b) Far-field pressure spectrum at $\theta = 30^\circ$; --- simulation; - - - measurements. 8

1.4 (a) Snapshots of the vorticity field: $\omega_z$ in the $x$ y plane at $z = 0$. (b) Overall sound pressure level as a function of angle $\theta$ measured from the jet axis, at 60$r_0$ from the jet nozzle. 9

1.5 Normalized centerline axial turbulence intensity (a) and its peak value in the shear-layer (b) with different co-flow velocities. 10

1.6 Main components of dual stream jet flow: Primary potential core length, $x_p$, generalized secondary core (GSC) length, $x_{GSC}$, and protrusion of inner nozzle, $x_{prot}$. 12

1.7 (a) Snapshots of the vorticity for coplanar dual jets with $M_p = 0.75$, $M_s = 0.85$ and $M_{CF} = 0.28$. (b) Computed and measured OASPL directivity spectra for coplanar dual jets in still air (red) and in co-flow (green). 13

1.8 (a) Twin GE CF6-50E2 turbo-fan engines mounted under the wing on a Boeing 747. (b) Structured mesh surface shown with round coaxial nozzle and pylon configuration. 15

1.9 (a) Mach number on symmetry plane of round nozzle with pylon (Config. 6) (b) Turbulent Kinetic Energy contours on symmetry plane of round nozzle with pylon (Config. 6) 15

1.10 Change in noise level due to pylon. 17
1.11 Instantaneous vorticity contours and different FW-H/Kirchhoff surfaces tested ........................................ 20
1.12 Effect of FW-H surface length on the 1/3-octave spectra at inlet angle $\theta = 90^\circ$, (a) Kirchhoff formulation (b) FW-H standard formulation (c) FW-H with $\rho^{\ast}$ substitution ........................................ 21
1.13 Vorticity field of flow past a cylinder computed from CFD ................................................................. 22
1.14 (a) Acoustic signals computed for various Kirchhoff integration surfaces as shown in figure 1.13, (b) Acoustic signals computed for various FW-H integration surfaces as shown in figure 1.13 ........................................ 23

2.1 $\alpha \Delta x$ vs $\bar{\alpha} \Delta x$ curves, .... 6th order central difference scheme, ___

2.2 Actual geometry boundary (blue) and computational marked boundary (green) used in immersed boundary method. Three points (black) show the three possibilities for a grid point .................................................. 46

2.3 Multi-block grid generated for dual-stream nozzle ................................................................. 51
2.4 Levels of communication between the processors ................................................................. 52
2.5 Centerline Grid Singularity ................................................................. 52
2.6 Wasistho buffer function is plotted for $C_2 = 10$ and $C_1 = 1.0$ in equation 2.78 ......................................................... 56
2.7 (a) FW-H integration surface showing two integration patches (blue and black dots) shared by one common row of points (red dots). (b) Location of the observer defined with respect to jet exit ......................................................... 57

2.8 Process flow-chart for the EAGLEJet RANS-LES solver and PS-FWH noise calculation solver ......................................................... 62

3.1 Solver performance: (a) Code speed up with different number of processors used on a 3D two block Gaussian pulse case, (b) Comparison of execution times for inefficient and optimal load balancing on processors. Total number of processors used is two ......................................................... 66

3.2 Initial pressure disturbance with Gaussian spatial distribution centered at $(x, y) = (0, 20)$ used to model the reflection of a acoustic pulse by a wall. The half-width of the pulse is 3.0 and $y = 0$ is the solid wall ......................................................... 66

3.3 (a) Numerical pressure contours at different time steps with a Gaussian pulse centered at $(0, 20)$ and with wall reflection from $y = 0$ plane, (b) Numerical and exact pressure contours at same time steps given in Tam and Dong ......................................................... 67
<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.4</td>
<td>Pressure waveforms of an acoustic pulse reflected off a solid wall along the line $x = y$ at different times. Exact solution shown by black line at $t/\Delta t = 1000$, other color lines show the numerical solution.</td>
</tr>
<tr>
<td>3.5</td>
<td>(a) $201 \times 201$ grid generated for the $5^\circ$ wedge. (b) Pressure contours in a steady-state simulation of Mach 1.5 flow around a $5^\circ$ wedge with an oblique shock angle of $50^\circ$.</td>
</tr>
<tr>
<td>3.6</td>
<td>Supersonic flow around $10^\circ$ wedge</td>
</tr>
<tr>
<td>3.7</td>
<td>Supersonic flow along $5^\circ$ wedge: dissipation effects</td>
</tr>
<tr>
<td>3.8</td>
<td>Gaussian pulse propagation in a 2D five block grid</td>
</tr>
<tr>
<td>3.9</td>
<td>Gaussian pulse propagation in a 2D four block grid</td>
</tr>
<tr>
<td>3.10</td>
<td>Gaussian pulse propagation in a 2D four block grid</td>
</tr>
<tr>
<td>3.11</td>
<td>Gaussian pulse propagation in a 3D two block grid</td>
</tr>
<tr>
<td>3.12</td>
<td>Comparison of exact and numerical 2D and 3D Gaussian pulse propagation</td>
</tr>
<tr>
<td>3.13</td>
<td>Developed turbulent boundary layer profile with Mach 0.2 flow along the flat plate of length $L = 5.09\ m$.</td>
</tr>
<tr>
<td>3.14</td>
<td>Turbulent boundary layer over flat plate</td>
</tr>
<tr>
<td>3.15</td>
<td>Pressure contours showing vortex shedding on the cylinder at $Re = 150$.</td>
</tr>
<tr>
<td>3.16</td>
<td>Time series plot of pressure showing lift and drag variation on the cylinder.</td>
</tr>
<tr>
<td>3.17</td>
<td>Non-matching grid and Mach contours for flow past cylinder at $M = 0.2$.</td>
</tr>
<tr>
<td>3.18</td>
<td>Variation of $C_p$ and $St$ for flow past cylinder at $M = 0.2$ using different turbulence models.</td>
</tr>
<tr>
<td>3.19</td>
<td>Mesh used for RAE2822 airfoil and the variation of pressure around the airfoil at $M = 0.729$.</td>
</tr>
<tr>
<td>3.20</td>
<td>Pressure coefficient variation on the transonic RAE2822 airfoil surface.</td>
</tr>
<tr>
<td>3.21</td>
<td>FW-H code validation using periodic pressure pulse.</td>
</tr>
<tr>
<td>4.1</td>
<td>(a) Boeing convergent nozzle geometry with $M_d = 1.0$ and $D_j = 2.45\ in.$ (b) Observer location in terms of polar distance $r$ and inlet angle $\theta$ with respect to the nozzle exit.</td>
</tr>
</tbody>
</table>
4.2 Mesh generated for convergent nozzle with \( D_j = 2.45 \) in: Computational domain shown with black lines. Two slices extracted at constant axial locations inside and outside the nozzle, total number of grid points in the computational domain = 5.0 million. Left inset: grid topology inside the nozzle with a H-type Cartesian grid in the center surrounded by four cylindrical blocks, Right inset: grid topology outside the nozzle further downstream.

4.3 Computational domain shown for a nozzle case. Boundary conditions shown on different surfaces and a typical grid FW-H surface shown in grey.

4.4 1/3-octave SPL at different inlet angles plotted for coarse (3.5 million) and fine grid (5.0 million) and experiments for \( M_j = 0.9 \) unheated jet.

4.5 OASPL (dB) at different inlet angles plotted for coarse (3.5 million) and fine grid (5.0 million) and experiments for \( M_j = 0.9 \) unheated jet.

4.6 A converging diverging (CD) nozzle operating at \( M_j = 0.5 \). (a) FW-H surfaces shown: \( S_1 \) to \( S_4 \) on non-dimensional pressure gradient (normalized with \( \rho_o, c_o \) and \( D_j \)) contours. (b) Comparison of 1/3-octave SPL obtained for FW-H surfaces \( S_1-S_4 \) at \( r/D = 72 \) and \( \theta = 150^\circ \).

4.7 (a) Iso-streams showing the mass entrainment for single stream nozzle without co-flow, \( M_j = 0.9 \) and \( TTR = 2.7 \). (b) Longitudinal evolution of the mean flow rate for single stream convergent nozzle \( M_j = 0.9 \) and \( TTR = 2.7 \).

4.8 Vorticity magnitude contours plotted for convergent jet (\( D_j = 2.45 \) in) operating at \( M_j = 0.9 \), \( TTR = 1.0 \), \( M_{cf} = 0.0, 0.12, 0.2 \) and 0.28. Slices extracted at different axial locations (0.015 < \( x/D_j < 10 \)) downstream of the nozzle exit to show the transition from laminar to turbulent flow in the shear layer.

4.9 (a) Iso-surface contours of \( Q \)-criterion plotted at \( Q = \pm 10^5 s^{-2} \) for single stream nozzle without co-flow, \( M_j = 0.9 \) and \( TTR = 1.0 \), (b) Centerline turbulence intensity level variation for single stream convergent nozzle \( M_j = 0.9 \) and \( TTR = 1.0 \), with parallel stream Mach numbers, \( M_{sf} = 0.0, 0.12, 0.2 \) and 0.28.

4.10 Iso-surfaces of \( Q \)-criterion plotted at \( Q = \pm 10^5 s^{-2} \) for the jet (\( D_j = 2.45 \) in) operating at \( M_j = 0.9 \), \( TTR = 1.0 \). (a) \( M_{cf} = 0.0 \). (b) \( M_{cf} = 0.12 \). (c) \( M_{cf} = 0.2 \). (d) \( M_{cf} = 0.28 \).

4.11 Turbulent Kinetic Energy (TKE) contours for convergent jet operating at \( M_j = 0.9 \), \( TTR = 1.0 \) and 2.7 and \( M_{sf} = 0.2 \).
4.12 Mean axial velocity contours for PIV measurements by Bridges ($D_j = 1 \text{ in}$) and numerical data ($D_j = 2.45 \text{ in}$): $M_j = 0.9$, $TTR = 1.0$, $M_{cf} = 0.0$.

4.13 Time averaged axial velocity profiles for convergent nozzle $D_j = 2.45 \text{ in}$ ($0.0622 \text{ m}$), $M_j = 0.9$ and $TTR = 1.0$, with parallel stream Mach numbers, (a) $M_{cf} = 0.0$, (b) $M_{cf} = 0.12$, (c) $M_{cf} = 0.2$ and (d) $M_{cf} = 0.28$.

4.14 Mean centerline (a) and radial (b) axial velocity profiles for single stream convergent nozzle $D_j = 2.45 \text{ in}$, $M_j = 0.9$ and $TTR = 1.0$, with parallel stream Mach numbers, $M_{cf} = 0.0, 0.12, 0.2$ and 0.28.

4.15 (a) Experimental and numerical values of the jet potential core length for different co-flow Mach numbers, $M_j = 0.9$ and $TTR = 1.0$, $M_{cf} = 0.0, 0.12, 0.2$ and 0.28. (b) Pressure gradient contours (normalized with $\rho_o, c_o$ and $D_j$) with FW-H acoustic surface used to sample data for noise calculations.

4.16 (a) Radial mean axial velocity profiles at different axial locations for convergent nozzle operating at $M_j = 0.9$, $TTR = 1.0$, $M_{cf} = 0.0$. (b) Boundary layers on the nozzle walls for $M_j = 0.9$, $TTR = 1.0$, $M_{cf} = 0.0$ to 0.28.

4.17 Displacement thickness ($\delta^*$) value on inner and outer nozzle walls for jet operating at $M_j = 0.9$, $M_{cf} = 0.0$ to 0.28. (a) $TTR = 1.0$ and (b) $TTR = 2.7$.

4.18 Mean axial velocity and temperature contours for cold and heated convergent nozzle $M_j = 0.9$ jet.

4.19 Centerline mean axial velocity (left y-axis) and centerline turbulence intensity levels (right y-axis) for single stream nozzle, $M_j = 0.9$ and $TTR = 1.0$ and 2.7, $M_{cf} = 0.0$ and 0.2.

4.20 Radial profiles of the relative temperature (a) and mean axial velocity (b) showing jet similarity inside and outside potential core region, $M_j = 0.9$ and $TTR = 2.7$. Radial distance normalized with half temperature radius $r_{1/2,TCL}$, half velocity radius $r_{1/2,UCL}$ and vorticity thickness $\delta_\omega$.

4.21 Radial profiles of the mean axial velocity and axial turbulence intensity at different axial locations. Radial similarity coordinate (equation 4.20) on x-axis calculated using jet half velocity radius and normalized with vorticity thickness calculated using equation 4.21. $CL$: centerline value at corresponding $x/D_j$.
4.22 Convergent nozzle $M_j = 0.9$ and $TTR = 1.0$ and 2.7: (a) Jet centerline relative temperature profile plotted against the axial distance normalized with the thermal potential core length defined as the axial location where temperature falls to 0.98 times its jet exit value, $M_{cf} = 0.0$ and 0.2. (b) Jet centerline axial turbulence intensity values plotted against the axial distance normalized with the dynamic potential core length defined as the axial location where axial velocity falls to 0.98 times its jet exit value.

4.23 Convergent nozzle $M_j = 0.9$ and $TTR = 1.0$ and 2.7: (a) Jet centerline mean axial velocity values plotted against the axial distance normalized with the dynamic potential core length. (b) Jet centerline mean axial velocity values plotted against the Witze parameter.

4.24 Convergent nozzle $M_j = 0.9$ and $TTR = 1.0$ and 2.7: Inverse of jet centerline mean axial velocity and radial location of jet half velocity values plotted against the axial distance.

4.25 Convergent nozzle $M_j = 0.9$ and $TTR = 1.0$ and 2.7: (a) Second order two point correlation coefficient for unheated jet. Red lines show the envelope joining the peak of correlation coefficients. (b) Two point correlation coefficient for heated jet. (c) Two point correlation coefficient for heated jet with $M_{cf} = 0.2$. (d) Fourth order two point correlation coefficient for unheated jet. (e) Fourth order two point correlation coefficient for heated jet. (f) Fourth order two point correlation coefficient for heated jet with $M_{cf} = 0.2$. $\xi$ is the separation distance between two probes. Correlations plotted in the shear layer $y/D_j = 0.5$ at $x/D_j = 5.0$.

4.26 Convergent nozzle $M_j = 0.9$ and $TTR = 1.0$ and 2.7: (a) Correlation coefficients at zero time delay for unheated jet (b) Correlation coefficients at zero time delay for heated jet. (c) Correlation coefficients at zero time delay for heated jet with $M_{cf} = 0.2$. (d) Variation of time delay for maximum cross correlation with separation distance for unheated jet. (e) Variation of time delay for maximum cross correlation with separation distance for heated jet. (f) Variation of time delay for maximum cross correlation with separation distance for heated jet with $M_{cf} = 0.2$. $\xi$ is the separation distance between two probes. Correlations plotted along the lip line $y/D_j = 0.5$ at $x/D_j = 5.0$. 

---

xv
4.27 Convergent nozzle $M_j = 0.9$ and $TTR = 1.0$ and 2.7: (a) Radial variation of flatness factor, $T_1(x)$ for unheated jet (b) Radial variation of flatness factor, $T_1(x)$ for heated jet. (c) Radial variation of flatness factor, $T_1(x)$ for heated jet with $M_{cf} = 0.2$. (d) Convection velocity as a function of $St$ number for unheated jet. (e) Convection velocity as a function of $St$ number for heated jet. (f) Convection velocity as a function of $St$ number for heated jet with $M_{cf} = 0.2$.

4.28 (a) Predicted and experimental 1/3-octave noise spectra plotted against $St$ for convergent nozzle, $M_j = 0.9$ and $TTR = 1.0$, $M_{cf} = 0.0$ at $r/D_j = 72$. (b) Predicted and experimental 1/3-octave noise spectra plotted against $St$ for convergent nozzle, $M_j = 0.9$ and $TTR = 2.7$, $M_{cf} = 0.0$ at $r/D_j = 72$.

4.29 Predicted and experimental 1/3-octave noise spectra plotted against $St$ for convergent nozzle, $\theta = 90^\circ$, $M_j = 0.9$ and $TTR = 1.0$.

4.30 (a) Numerical 1/3-octave noise spectra for different inlet angles at $r/D_j = 72$ plotted against $St$ for convergent nozzle without parallel stream, $M_j = 0.9$ and $TTR = 1.0$ and 2.7. (b) Predicted OASPL at different inlet angles for $TTR = 1.0$ and 2.7 and $r/D_j = 72$.

4.31 Convergent nozzle operating at $M_j = 0.9$, $M_{cf} = 0.2$, and $TTR = 1.0$ and 2.7: (a) Numerical 1/3-octave noise spectra for different inlet angles at $r/D_j = 72$ plotted against $St$. (b) Predicted OASPL at different inlet angles at $r/D_j = 72$.

4.32 Flight velocity coefficient obtained in experiments: (a) Unheated $M_j = 0.9$ jet, tunnel Mach number $M_t = 0.048-0.20$. Data also shown from NGTE wind tunnel experiments. Tanna and Morris and (b) SAE ARP876 Report, 1994.

4.33 (a) Numerical prediction of 1/3-octave noise spectra plotted against $St$ for convergent nozzle, $M_j = 0.9$ and $TTR = 1.0$, for different co-flow Mach numbers $M_{cf} = 0.0$, 0.12, 0.2 and 0.28 at $r/D_j = 72$. (b) Comparison of OASPL at different inlet angles for single stream nozzle, $M_j = 0.9$ and $TTR = 1.0$, for different co-flow Mach numbers $M_{cf} = 0.0$ to 0.28 at $r/D_j = 72$.

4.34 Least square fit between reduction in the OASPL and different co-flow speeds for $\theta = 150^\circ$ to 50$^\circ$ for single stream nozzle, $M_j = 0.9$, $TTR = 1.0$, $M_{cf} = 0.0$ to 0.28.

4.35 (a) Numerical variation of flight velocity exponent $m$ for different inlet angles, $M_j = 0.9$, $TTR = 1.0$. (b) Experimental variation of the flight velocity exponent for different inlet angles given in Viswanathan and Czech.
4.36 Jet noise spectra scaling for unheated jet with \( M_j = 0.9, TTR = 1.0, M_{cf} = 0.0 \) and 0.12. ........................................... 133

5.1 Multi-block grid topology inside and outside the dual-stream nozzle with \( D_{pj} = 2.45 \text{ in} \) (0.0622 m) and \( D_{sj} = 6.00 \text{ in} \) (0.15 m). (a) Cylindrical mesh surrounding the H-type mesh in the center to remove centerline singularity. (b) Cylindrical mesh surrounding the dual-stream nozzle. ........................................... 138

5.2 Computational domain for dual-stream jet simulation: Black lines show the axial and radial extent of the computational domain. The primary nozzle geometry is shown in blue surrounded by secondary nozzle in red. Inset (left) shows a slice extracted inside the nozzles. A cylindrical mesh surrounds the H-type mesh in the center to remove centerline singularity. Inset (right) shows the grid at a slice at a downstream location outside the nozzle. ............................... 138

5.3 (a) Grid clustering to resolve primary and secondary shear layers in the dual-stream nozzle. (b) Observer location specified in terms of polar distance \( r \) and nozzle inlet angle \( \theta \) with respect to the core nozzle exit. ............................................................ 139

5.4 Mean axial velocity and temperature contours for dual-stream nozzle \( NPR_p = 1.6, TTR_p = 1.0, NPR_s = 1.8, TTR_s = 1.0 \), with parallel stream Mach numbers, \( M_{cf} = 0.0 \) and 0.2: (a) \( M_{cf} = 0 \), (b) \( M_{cf} = 0.2 \). Instantaneous pressure perturbation contours: (c) \( M_{cf} = 0 \), (d) \( M_{cf} = 0.2 \). .................................................. 140

5.5 Mean axial velocity and instantaneous pressure contours for different coflow Mach numbers for dual-stream jet operating at \( NPR_p = 1.6, TTR_p = 2.26, NPR_s = 1.8, TTR_s = 1.0, M_{cf} = 0.0, 0.2, 0.28 \). 141

5.6 (a) (d) Time averaged temperature (\( K \)) and TKE contours for dual-stream nozzle \( NPR_p = 1.6, TTR_p = 2.26, NPR_s = 1.8, TTR_s = 1.0, M_{cf} = 0.0 \) and 0.2. (e) Instantaneous density contours normalized with free-stream \( \rho_{ref} \) for \( TTR_p = 2.26 \) and \( M_{cf} = 0.2 \). .................................................. 142

5.7 Boundary layers development on the walls of the (a) (c) core and (b) (d) fan nozzles for dual-stream convergent nozzle operating at \( NPR_p = 1.6, TTR_p = 1.0 \) and 2.26, \( NPR_s = 1.8, TTR_s = 1.0, M_{cf} = 0.0 \) and 0.2. .................................................. 143

5.8 Displacement thickness (\( \delta^* \)) values on (a) (c) inner and (b) (d) outer surfaces of the core and fan nozzles for dual-stream jet \( NPR_p = 1.6, TTR_p = 1.0 \) and 2.26, \( NPR_s = 1.8, TTR_s = 1.0 \), with parallel stream Mach numbers, \( M_{cf} = 0.0 \) and 0.2 .................................................. 145
5.9 Variation of centerline mean axial velocity and centerline and shear layers turbulence intensity values along the jet axis for jet with co-flow $M_{cf} = 0.0$ and 0.2, $NPR_p = 1.6$, $NPR_s = 1.8$, $TTR_s = 1.0$:
(a) $TTR_p = 1.0$, (b) $TTR_p = 2.26$, (d) Variation of mean centerline temperature values along the jet axis for jet operating at $NPR_p = 1.6$, $TTR_p = 2.26$, $NPR_s = 1.8$, $TTR_s = 1.0$ with co-flow $M_{cf} =$ 0.0 and 0.2.

5.10 Vorticity magnitude contours plotted for dual-stream jet operating at $M_{pj} = 0.85$, $M_{sj} = 0.95$, $TTR_s = 1.0$, $M_{cf} = 0.0$ and 0.2. Slices extracted at different axial locations ($0.015 < x/D_{pj} < 10$) downstream of the nozzle exit to show the transition from laminar to turbulent flow in the shear layer. Both streams merge to form an uniform mixing region downstream of the nozzle. (a) $TTR_p = 1.0$, $M_{cf} = 0.0$, (b) $TTR_p = 1.0$, $M_{cf} = 0.2$, (c) $TTR_p = 2.26$, $M_{cf} = 0.0$, (d) $TTR_p = 2.26$, $M_{cf} = 0.2$.

5.11 Radial profiles of the relative temperature (a) and mean axial velocity (b) showing jet similarity inside and outside potential core region, $TTR_p = 2.26$, $M_{cf} = 0.2$. Radial distance normalized with half temperature radius and half velocity radius respectively.

5.12 Iso-surfaces of $Q$ criterion ($Q = \pm 5 \times 10^3$ $s^{-2}$) for dual-stream nozzle operating at $NPR_p = 1.6$, $TTR_p = 1.0$, $NPR_s = 1.8$, $TTR_s = 1.0$, $M_{cf} = 0$. Pressure perturbations shown on the enclosing acoustic surface.

5.13 Iso-surfaces of $Q$ criterion ($Q = \pm 10^6$ $s^{-2}$) for dual-stream nozzle operating at $NPR_p = 1.6$, $TTR_p = 2.26$, $NPR_s = 1.8$, $TTR_s = 1.0$: (a) $M_{cf} = 0.0$, (b) $M_{cf} = 0.0$ and 0.2.

5.14 Probes (shown by red circles) along the primary and secondary shear layers for the calculation of the two-point correlations in the axial locations. Reference probes at $x/D_{pj} = 2$ and 3 for the primary and secondary shear layers respectively.

5.15 Dual-stream nozzle operating at $M_p = 0.85$, $M_s = 0.95$, $TTR_s = 1.0$, and $M_{cf} = 0.0$ and $TTR_p = 1.0$ and 2.26: Second order two point correlation coefficients plotted along the primary shear layer for (a) $TTR_p = 1.0$, $M_{cf} = 0.0$, (b) $TTR_p = 2.26$, $M_{cf} = 0.0$, (c) $TTR_p = 2.26$, $M_{cf} = 0.2$. Fourth order two point correlation coefficients plotted along the primary shear layer for (d) $TTR_p = 1.0$, $M_{cf} = 0.0$, (e) $TTR_p = 2.26$, $M_{cf} = 0.0$, (f) $TTR_p = 2.26$, $M_{cf} = 0.2$. $\xi$ is the separation distance between two probes. Correlations plotted in the shear layer $y/D_{pj} = 0.5$ at $x/D_{pj} = 2.0$.
5.16 Dual-stream nozzle operating at $M_p = 0.85$, $M_s = 0.95$, $TTR_s = 1.0$, and $M_{cf} = 0.0$ and $TTR_{p} = 1.0$ and 2.26: Second order two point correlation coefficients plotted along the secondary shear layer for
(a) $TTR_{p} = 1.0$, $M_{cf} = 0.0$, (b) $TTR_{p} = 2.26$, $M_{cf} = 0.0$, (c) $TTR_{p} = 2.26$, $M_{cf} = 0.2$. Fourth order two point correlation coefficients plotted along the secondary shear layer for
(d) $TTR_{p} = 1.0$, $M_{cf} = 0.0$, (e) $TTR_{p} = 2.26$, $M_{cf} = 0.0$, (f) $TTR_{p} = 2.26$, $M_{cf} = 0.2$. $\xi$ is the separation distance between two probes. Correlations plotted in the shear layer $y/D_{pj} = 1.0$ at $x/D_{pj} = 3.0$.

5.17 Dual-stream nozzle operating at $M_p = 0.85$, $M_s = 0.95$, $TTR_s = 1.0$, and $TTR_{p} = 1.0$ and 2.26: (a) Correlation coefficients at zero time delay for unheated core jet, (b) Correlation coefficients at zero time delay for heated core jet, (c) Correlation coefficients at zero time delay for heated core jet with $M_{cf} = 0.2$, (d) Variation of time delay for maximum cross correlation with separation distance for unheated core jet, (e) Variation of time delay for maximum cross correlation with separation distance for heated core jet, (f) Variation of time delay for maximum cross correlation with separation distance for heated core jet with $M_{cf} = 0.2$. $\xi$ is the separation distance between two probes. Correlations plotted along the primary lip line $y/D_{pj} = 0.5$ at $x/D_{pj} = 2.0$.

5.18 Dual-stream nozzle operating at $M_p = 0.85$, $M_s = 0.95$, $TTR_s = 1.0$, and $TTR_{p} = 1.0$ and 2.26: (a) Correlation coefficients at zero time delay for unheated core jet, (b) Correlation coefficients at zero time delay for heated core jet, (c) Correlation coefficients at zero time delay for heated core jet with $M_{cf} = 0.2$, (d) Variation of time delay for maximum cross correlation with separation distance for unheated core jet, (e) Variation of time delay for maximum cross correlation with separation distance for heated core jet, (f) Variation of time delay for maximum cross correlation with separation distance for heated core jet with $M_{cf} = 0.2$. $\xi$ is the separation distance between two probes. Correlations plotted along the secondary lip line $y/D_{pj} = 1.0$ at $x/D_{pj} = 3.0$.
5.19 Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 2.26$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$. (a) Flatness ($T_1(x)$) parameter along the radial direction at $x/D_{pj} = 5.0$ for $M_{cf} = 0.0$, $TTR_p = 1.0$. (b) Flatness ($T_1(x)$) parameter along the radial direction at $x/D_{pj} = 5.0$ for $M_{cf} = 0.2$, $TTR_p = 2.26$. (c) Flatness ($T_1(x)$) parameter along the radial direction at $x/D_{pj} = 5.0$ for $M_{cf} = 0.2$, $TTR_p = 2.26$. Variation of phase velocity as a function of Strouhal number (d) $TTR_p = 1.0$, $M_{cf} = 0.0$, (e) $TTR_p = 2.26$, $M_{cf} = 0.0$, (f) $TTR_p = 2.26$, $M_{cf} = 0.2$. 159

5.20 Location of the FW-H surface for a dual-stream nozzle plotted against pressure gradient (normalized with $\rho_o$, $c_o$ and $D_{pj}$). 161

5.21 Experimental and numerical 1/3-octave noise spectra compared at different nozzle inlet angles at 122$D_e$ for dual-stream nozzle with $NPR_p = 1.6$, $TTR_p = 2.26$, $NPR_s = 1.8$, $TTR_s = 1.0$, $M_{cf} = 0.0$. Equivalent jet diameter $D_e = 4.9$ in. Strouhal number calculated using primary jet flow parameters. 162

5.22 Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 1.0$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, $M_{cf} = 0.0$ ans 0.2. Equivalent jet diameter $D_e = 4.9$ in. Strouhal number calculated using primary jet flow parameters, (a) 1/3-octave $SPL$ compared at different nozzle inlet angles at 122$D_e$. (b) Variation of overall noise levels (OASPL) with different nozzle inlet angles at 122$D_e$ with $M_{cf} = 0.0$ and 0.2. 163

5.23 Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 1.0$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, $M_{cf} = 0.0$ ans 0.2. Equivalent jet diameter $D_e = 4.9$ in. Strouhal number calculated using primary jet flow parameters. (a) 1/3-octave $SPL$ compared at different nozzle inlet angles at 122$D_e$. (b) Variation of overall noise levels (OASPL) with different nozzle inlet angles at 122$D_e$ with $M_{cf} = 0.0$ and 0.2. 164

5.24 Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 1.0$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, $V_{pj} = 291$ m/s, $V_{mix} = 315.0$ m/s. (a), (b) Least squares fit curves for $\Delta OASPL$ vs. $M_{cf}$ at different inlet angles using $V_{ref} = V_{pj}$. (c) Variation of flight velocity exponent with nozzle inlet angles using $V_{ref} = V_{pj}$. (d) Least squares fit curves for $\Delta OASPL$ vs. $M_{cf}$ at different inlet angles using $V_{ref} = V_{mix}$. (e) Variation of flight velocity exponent with nozzle inlet angles using $V_{ref} = V_{mix}$. 165
5.25 Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 2.26$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, $V_{pj} = 408.0\ m/s$, $V_{mix} = 334.0\ m/s$. (a), (b) Least squares fit curves for $\Delta OASPL$ vs. $M_{cf}$ at different inlet angles using $V_{ref} = V_{pj}$. (c) Variation of flight velocity exponent with nozzle inlet angles using $V_{ref} = V_{pj}$. (d), (e) Least squares fit curves for $\Delta OASPL$ vs. $M_{cf}$ at different inlet angles using $V_{ref} = V_{mix}$. (f) Variation of flight velocity exponent with nozzle inlet angles using $V_{ref} = V_{mix}$.

5.26 Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 1.0$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, $V_{pj} = 291.0\ m/s$, $V_{mix} = 315.0\ m/s$. (a) Jet noise spectra scaling using $V_{pj}$. (b) Jet noise spectra scaling using $V_{mix}$. $r/D_e = 122$. Black circles: $M_{cf} = 0$, Red triangles: $M_{cf} = 0.2$.

5.27 Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 2.26$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, $V_{pj} = 408.0\ m/s$, $V_{mix} = 334.0\ m/s$. (a) Jet noise spectra scaling using $V_{pj}$. (b) Jet noise spectra scaling using $V_{mix}$. $r/D_e = 122$. Black circles: $M_{cf} = 0$, Red triangles: $M_{cf} = 0.2$.

5.28 Grid generation for dual-stream nozzle with pylon $D_{pj} = 2.45\ in$ and $D_{sj} = 6.1\ in$. (a) Dual-stream nozzle showing simplified pylon structure, (b) One half computational domain showing block boundaries and pylon structure (solid red lines).

5.29 Grid for nozzle with pylon showing the multi-block structured grid with grid clustering performed in different regions.

5.30 Dual-stream with pylon jet operating at $M_p = 0.85$, $M_s = 0.95$, $TTR_p = 1.0$, $TTR_s = 1.0$ and $M_{cf} = 0.12$. Instantaneous Mach contours (a) in pylon plane, (b) perpendicular to the pylon plane. Vorticity contours (c) in pylon plane, (d) perpendicular to the pylon plane.

5.31 Jet operating at $M_p = 0.85$, $M_s = 0.95$, $TTR_p = 1.0$, $TTR_s = 1.0$ and $M_{cf} = 0.12$: Mean axial velocity contours (a) in the pylon plane, (b) out of the pylon plane.

5.32 Vorticity contours near the pylon end showing flow circulation.

5.33 Mean axial velocity slices at different axial locations showing asymmetric nature of the jet plume.

B.1 Transformation of the interpolation from the physical domain to the computational domain at the non-matching block interfaces

D.1 Flowchart showing the directory and file layout for the code
## List of Tables

1.1 High-bypass turbofan engines with the commercial aircraft on which they are installed. ........................................ 11
2.1 Coefficients for five stage LDDRK scheme ......................... 39
2.2 Coefficients for 6th order low pass filter, \(-0.5 \leq \alpha_f \leq 0.5\) ............... 44
3.1 Parallel Solver Speedup ........................................... 64
3.2 3D Gaussian pulse case: Load balancing, total number of processors used is 2. ........................................... 65
3.3 Flow conditions for turbulent flat plate with length \(L = 5.09\ m\) ....... 75
3.4 Freestream conditions for flow past RAE2822 airfoil ..................... 82
4.1 Convergent nozzle jet operating conditions, \(\lambda = M_{cf}/M_j\) .......... 86
4.2 FW-H acoustic surface axial and radial closure locations .................. 94
4.3 Comparison of numerical potential core length values with empirical formula for single stream nozzle \(D_j = 2.45\ in\), and different co-flow Mach numbers, \(\lambda = M_{cf}/M_j\) ........................................ 103
4.4 Comparison of jet flow characteristics for unheated and heated single stream nozzle \(D_j = 2.45\ in\), \(M_j = 0.9\), \(TTR = 1.0\) and 2.7, \(T_j = 680.81\ K\) and \(M_{cf} = 0.0, 0.12, 0.2\) and 0.28 ........................................ 109
4.5 Predicted length and time scales, and convection speed of the turbulent structures of the cold and heated jets operating at \(M_j = 0.9\). Calculated at \(x/D_j = 5.0\) along the lip line based on second order cross-correlation of the axial velocity fluctuations ................... 123
5.1 Dual-stream jet operating conditions, \(D_{pj} = 2.45\ in, D_{sj} = 6.00\ in\) ... 136
5.2 Comparison of jet flow characteristics for dual-stream jet operating at \(M_p = 0.85\), \(NPR_p = 1.6\), \(TTR_p = 1.0\) and 2.26, \(M_s = 0.95\), \(NPR_s = 1.8\), \(TTR_s = 1.0\), and \(M_{cf} = 0.0\) and 0.2 ............... 150
5.3 Predicted length and time scales, and convection speed of the turbulent structures of dual-stream jets ($TTR_p = 1.0$ and 2.26) operating with and without co-flow. Calculated at $x/D_{pj} = 2.0$ on the core lip line and at $x/D_{pj} = 3.0$ on the fan lip line based on cross-correlation of the axial velocity fluctuations.

<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>D.1</td>
<td>Boundary conditions implemented in the code (* not to be set by user)</td>
</tr>
<tr>
<td></td>
<td>D.2 Initial conditions options implemented in the code.</td>
</tr>
</tbody>
</table>

160

219

229
List of Symbols

**Symbols**

ω \quad \text{vorticity magnitude}

ξ, η, ζ \quad \text{orthogonal coordinate system in computational domain}

δ \quad \text{Kronecker’s Delta}

δ_D \quad \text{Diameter ratio, } D_i/D_p

γ \quad \text{ratio of specific heats, } \gamma = 1.4 \text{ for air}

λ \quad \text{spectral radius of the flux Jacobian matrix}

λ_A, λ_V \quad \text{Area and velocity ratio (secondary divided by primary)}

λ_T \quad \text{Static temperature ratio (primary divided by secondary)}

μ \quad \text{molecular or dynamic viscosity}

ν \quad \text{kinematic viscosity}

ω \quad \text{angular frequency}

ρ \quad \text{density}

σ \quad \text{CFL number}

τ \quad \text{viscous stress tensor}

θ \quad \text{nozzle inlet angle}

ε \quad \text{average rate of dissipation of turbulence kinetic energy per unit mass}
\( \eta_K \)  
Kolmogorov length scale

\( \Delta t \)  
fictitious time step

\( \Delta \tau \)  
physical time step

\( c \)  
speed of sound

\( C_{1111} \)  
fourth order cross-correlation value

\( D \)  
nozzle exit diameter

\( D_j \)  
diameter of fully expanded jet

\( e, e_{\text{tot}} \)  
total energy per unit mass, 
\[
 e = \frac{p}{\gamma - 1} + \frac{1}{2} \rho (u^2 + v^2 + w^2) 
\]

\( E, F, G \)  
inviscid flux terms in the governing equations

\( E_V, F_V, G_V \)  
viscous flux terms in the governing equations

\( f \)  
frequency

\( G \)  
amplification factor in the von Neumann stability analysis

\( k \)  
wavenumber

\( TKE \)  
turbulent kinetic energy, 
\[
 TKE = 0.5 (u'^2 + v'^2 + w'^2) 
\]

\( L \)  
length

\( m \)  
forward flight velocity exponent

\( M_c \)  
convective Mach number

\( M_d \)  
design Mach number

\( M_j \)  
jet Mach number

\( M_a \)  
acoustic Mach number

\( P \)  
pressure

\( \text{Pr} \)  
Prandtl number

\( Q \)  
flow variables array in conservative form

\( q \)  
heat transfer term in energy equation
R  gas constant of air at STP
$R_{11}$  second order cross-correlation value
$R_{1111}$  fourth order cross-correlation value
$Re$  Reynolds number, $Re = U_jD_j/\nu$
$r$  radial distance from the nozzle exit
$St$  Strouhal number
$T$  temperature
$u, v, w$  axial, vertical and spanwise velocity components in jet plume
$U_j$  fully expanded jet exit velocity
$U_c$  overall convection speed of the turbulent motions
$(x, y, z)$  three directions in Cartesian coordinates
$<>$  mean value

**Subscript**

$s$  secondary(fan) stream
$\infty$  reference values
$i, j, k$  indices used to represent three orthogonal directions in the equations
$p$  primary(core) stream
$0$  total value

**Superscript**

$'$  perturbation part of a variable around its time-averaged value

**Abbreviations**

$BPR$  bypass ratio
$CAA$  computational aero-acoustics
$CDN$  convergent divergent nozzle
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>DES</td>
<td>detached-eddy simulation</td>
</tr>
<tr>
<td>DNC</td>
<td>direct noise computation</td>
</tr>
<tr>
<td>DRP</td>
<td>dispersion relation preserving</td>
</tr>
<tr>
<td>FW-H</td>
<td>Ffowcs Williams and Hawkings</td>
</tr>
<tr>
<td>IBM</td>
<td>immersed boundary method</td>
</tr>
<tr>
<td>LDDRK</td>
<td>low dissipation and low dispersion Runge-Kutta method</td>
</tr>
<tr>
<td>LES</td>
<td>large eddy simulation</td>
</tr>
<tr>
<td>MILES</td>
<td>monotonically integrated large eddy simulation</td>
</tr>
<tr>
<td>NPR</td>
<td>nozzle pressure ratio</td>
</tr>
<tr>
<td>OASPL</td>
<td>overall sound pressure level</td>
</tr>
<tr>
<td>SGS</td>
<td>sub-grid scales</td>
</tr>
<tr>
<td>TTR</td>
<td>ratio of total temperature to the free-stream temperature</td>
</tr>
</tbody>
</table>
Acknowledgments

First and foremost, I would sincerely like to thank my advisor Prof. Philip J. Morris for his guidance, insight, and continuing support through my master's and doctoral study at Penn State. Throughout my PhD, his valuable suggestions have enabled me to advance my research apart from meeting the project deadlines. His other professional qualities such as hard work and punctuality, to name a few, have been an equally integral learning experience.

I would like to acknowledge the support for my PhD project from Pratt & Whitney (Dr. Constantine Baltas, contract monitor). Sincere thanks to Dr. Constantine Baltas for his continuous support and valuable remarks during our monthly meetings. I would also like to thank Dr. Fuhua Ma for his cooperation and help in successfully transferring and running the EAGLEJet solver at Pratt & Whitney servers.

I am grateful to Dr. Kenneth Brentner for providing the computational facilities. I would like to thank my doctoral committee members Profs. Dennis McLaughlin, Victor Sparrow and Kenneth Brentner for their constructive criticisms and useful suggestions. I am grateful to Dr. K. Viswanathan for providing noise data from Boeing and Dr. James Bridges for providing the flow and noise data which has been very useful in validating the present results. My special thanks to Mr. Kirk Heller for always helping me with the server issues and patiently answering my endless questions regarding the computational facilities. I would like to acknowledge the promptness and helpful nature of the staff of the Aerospace Engineering.

I am grateful to James Erwin for giving me the NLDE solver which formed the basis for the current EAGLEJet solver. My special thanks to Dr. Yongle Du for helping me at several stages of my research. I would like to thank my colleagues Dr. Steven A. E. Miller, Jogesh Chopra, Monica Christiansen, and all other lab-mates for useful discussions and making my work life comfortable at Penn State.
I am grateful for the encouragement and support I received from Zonta Foundation and would like to thank them for awarding me the Amelia Earhart Fellowship. Zonta Foundation is one of the very few organizations that encourage women to pursue career in Aerospace Sciences. I would like to acknowledge the contribution of all my teachers, and the facilities I was fortunate to receive at Indian Institute of Technology, Kanpur that prepared me for my graduate studies.

I would like to thank my roommates Dr. Meenakshi, Dr. Akansha and Sarmishtha and my friends Rakesh, Santhosh, Anusha, Aman, Hemant, Manu, Mohan, Manisha, Karthik and Pankaj to name a few, for making my life comfortable in State College. Special thanks to Dr. Nel Kopp for being a special part of my life at Penn State.

I would also like to thank my special friends Santosh, Jayam and Vaishali for their unconditional love and support throughout my life. I would like to warmly thank my parents Ashok Saxena and Kavita Saxena and my sisters Sonali and Shefali for their unconditional love and for standing by my decisions. My love and respect to my grandparents Mrs. Anjani Devi Saxena and Mr. Avadh Narayan Saxena for enriching my childhood and for giving me the teachings that will help me throughout my life.

Finally, I would like to acknowledge the numerous facilities and clubs at the Penn State campus that has helped me in nurturing my mind and soul.
The turbulent jet noise problem still remains one of the most complicated and difficult problems in aeroacoustics. Jet noise is an area of increasing importance as pressures for quieter aircraft increase. The current stringent aircraft noise regulations imposed by the Federal Aviation Administration (FAA) (Federal Aviation Regulation 36) and other international agencies such as International Civil Aviation Organization, have further intensified the need to perform accurate jet noise calculations for a more reliable estimation of the jet noise source. Realistic estimates of the noise produced due to installed aircraft engines will help the aviation industry in the development of noise reduction methods. One of the major contributors to the noise generated from the aircraft on take-off is the engine exhaust. Other sources of sound from jet engines include fan noise, combustion noise (low frequency and non-directional), internal engine components such as turbine, struts and splitters (peaks around 60° from downstream jet axis), instabilities in the mean jet flow and forward flight effects. The noise generated by an aircraft in the approach configuration has two main contributions, firstly the airframe noise resulting from turbulent flows over solid structures like wings, slats, flaps and landing gears and, secondly the engine noise generated from jet and fan flows. The latter propagates in both the upstream and the downstream directions. The dominant component of jet noise is due to the mixing of large-scale turbulent eddies. The noise characteristics of supersonic jets are very different from that of subsonic jets.
Supersonic jets operating at off-design conditions contain distinct components of turbulent mixing, screeching and broadband shock associated noise.

The theoretical formulation for the noise calculation, which forms the basis of the current research, was published in 1952 when Sir James Lighthill[1, 2] first developed a jet noise theory based on an acoustic analogy. He rearranged the mass and momentum conservation equations to construct a wave operator on the left hand side and all the terms left on the right hand side were treated as noise sources. These noise sources constitute the rate of mass change, unsteady force terms and nonlinear volume sources. The linear terms are associated with propagation effects that can then be determined as part of the solution. The non-linear terms are treated as “known” source functions to be determined by modeling and, in more recent approaches, parameterized with the parameters being determined from a steady RANS calculation. Since that time, jet noise research has come a long way with several important milestones. An important milestone achieved as a result of the Lighthill’s analogy was to supply the dimensional law of the radiated acoustical power as a function of the jet velocity $U_j$, the famous $U_j^8$ law for subsonic jets (See equation 2.49 on page 137 of Goldstein[3]). The power emission per unit length scales as $U_j^8$ up to about $8D_j$ downstream and falls very rapidly to zero after $8D_j$. The Lighthill power law reduces to $U_j^3$ instead of $U_j^8$ for supersonic jets at very high Mach numbers, though this result has been questioned recently. The directional pattern of the jet noise is the result of the convection Doppler factor $(1 - M_c \cos \theta)^{-5}$ which arises from the motion of the turbulent eddies relative to the observer. This assertion is also still the subject of controversy.

Lighthill’s acoustic analogy equation was generalized by Ffowcs Williams and Hawkings (FW-H) in 1969[4] when they included the effect of sound convection and moving surfaces in the famous FW-H acoustic analogy formulation and also introduced the use of Green’s function and generalized function theory[5] in solving the equation. Three major approaches are generally used to solve the FW-H equation: the retarded time formulation, the collapsing sphere and the emission time approach. Another major improvement in the acoustic analogy theory was made by Lilley[6] in 1974 when he used the linearized Euler equations to correct the wave propagation operator, which also explained the mean flow refraction ef-
fect and "cone of silence" in jets. This approach was used in the seventies and early eighties as a main component of jet noise research. Several experiments were conducted to study the noise generation mechanisms in jet flows and comparisons were made with the existing acoustic analogies.

Brown and Roshko in 1974 and others discovered that the turbulence in jets and free shear layers is made up of large turbulence structures as well as fine scale turbulence. The large scale structures dominate the overall mixing processes of jets and are important jet noise sources. This was an important discovery for jet noise research. Since then, an emphasis has been placed to account for the large turbulence scales in the jet flow in both empirical models and flow-field simulations from first principles.

Experiments by Troutt and McLaughlin have demonstrated that the Mach wave radiation that is created by the large scale structures dominates the sound field. In imperfectly expanded supersonic jets, two additional components of noise arise: screech tones and broadband shock associated noise. Figure 1.1(b) shows different noise components in an imperfectly expanded supersonic jet. On the other hand, subsonic jet noise is mainly due to turbulent mixing and has a uniform broadband spectrum as shown in figure 1.1(a). Tam compared the measured data of a subsonic jet (Ahuja) with the spectra for the large scale similarity and fine scale similarity spectral shapes and found good agreement between the two. This agreement provided strong evidence that the origin of noise in subsonic jets is also due to the large scale and fine scale turbulence structures. Since then many statistical models such as Tam and Chen have been proposed to describe large scale structures in free shear layers and jet flows.

With the development of several turbulence models and increases in computer power, it became possible to calculate turbulent jet noise from first principles. The most accurate method to simulate the turbulent flow is the Direct Numerical Simulation (DNS) that resolves all the turbulence scales ranging from largest scales (dependent on the flow conditions) to the Kolmogorov scales in the viscous regions ($\eta_K = (\nu^3/\epsilon)^{1/4}$). The grid requirements scale as $Re^{9/4}$ which escalate as Reynolds number increases for realistic nozzle operating conditions. DNS does not involve
Figure 1.1. (a) Typical subsonic broadband jet noise spectra $SPL$ vs. $St$ for a $M_j = 0.9$ jet [12]. (b) Far-field narrow-band noise spectra of a supersonic jet operating at $M_j = 2.0$ showing turbulent low frequency mixing noise, screeching tone and broadband shock associated noise [13]. Microphone at $30^\circ$ to nozzle inlet direction.
any modeling and is the most accurate method to study noise sources in a jet flow. However, due to the wide range of length and time scales present in turbulent flows and limited computing power, DNS is still restricted to low $Re$ number ($\sim 10,000$) flows in relatively simple geometries. Turbulence scales still have to be modeled in a way to perform simulation for problems of practical interest such as flows at realistic Reynolds number which may reach up to few millions for jet cases and for complicated jet geometries involving chevrons and pylon.

Another technique for modeling turbulence is Large Eddy Simulation (LES). In this method the large scales are resolved, and the effect of the unresolved small scales or the sub-grid scales (SGS) on the large scales is modeled using an SGS model. Large scales are generally much more energetic than the small ones and are directly affected by the boundary conditions. The small scales, however, are usually much weaker and tend to have a more or less universal character. Hence, it makes sense to simulate the more energetic large scales directly and model the effect of the small scales in jet flows. Noise generation is an unsteady process, which makes LES a powerful computational tool to be used in jet noise research since it can obtain time-accurate unsteady flow data. The Smagorinsky [14] model is a popular SGS model used in LES simulations. LES tends to become very expensive near walls as the resolution of the boundary layer needs very fine grid spacing. A surface integration method such as FW-H can be used to calculate the noise from the LES data. A review of the current status of this method being used for jet noise prediction is given in Bodony and Lele[15].

One of the other methods to make jet noise predictions is to use acoustic analogies in conjunction with the Reynolds averaged Navier-Stokes (RANS) equation solutions, with standard turbulence closures (See Wilcox[16]), for the mean flow. RANS models involve a lot of empiricism and hence will dissipate the large scale turbulent structures necessary for accurate noise calculations in subsonic jets when run as unsteady RANS models.

Several other hybrid methods, such as detached eddy simulation (DES) which is a combination of RANS and LES, have been proposed to benefit from the traditional modeling approaches of turbulent flow. These methods involve simulating the
RANS flow near the walls which switches to LES mode away from the walls to capture the unsteady large turbulent eddies. This method has been used recently for jet noise simulations by Shur et al. [17].

The current research involves noise predictions and analysis of high subsonic single and dual-stream jets including a pylon in commercial aircraft engines. A hybrid-RANS/LES turbulence modeling approach has been used in the current work and a parallel CFD solver has been developed to calculate the unsteady jet flow. The details of this method are given in chapter 2. The flow data is sampled on a FW-H acoustic surface wrapped around the jet plume and an acoustic solver based on solutions to the FW-H equation is used to calculate the far-field noise radiated by the jets.

The following subsections of this Chapter provide: a literature review of jet noise; the differences between the single stream and dual stream flow and noise characteristics; jet installation devices; the objectives of the current research; the numerical issues related to aeroacoustic computations; and an outline of the dissertation.

1.1 Literature Review: Subsonic Jet Noise

1.1.1 Single-Stream Jets

There have been several studies to predict jet noise that involve turbulent flow and far-field acoustic calculations. The first three-dimensional jet flow simulations with sound prediction appeared in the late 1990’s with the first DNS calculation conducted by Freund [18] for a $M = 0.9$ jet with $Re = 3600$ based on jet speed and exit diameter. A direct calculation of the noise was performed and the Lighthill’s acoustic analogy was also used to calculate the noise sources. The grid points reached 25 million in this calculation. Figure 1.2 shows the directivity of the far-field noise in a jet flow. The noise levels peak near $\theta = 30^\circ$ with angle measured from the downstream jet axis. Tam [9] showed that this highly directional noise component was generated by the large turbulence structures of the jet flow. Tam also observed that for $\theta$ greater than $70^\circ$ (see figure 1.2) the jet noise radiation was
Figure 1.2. Visualization of far-field sound: Velocity Gradients $\Theta = \nabla \cdot u$ in a round subsonic $M = 0.9$ jet calculated by solving the linear wave equation using DNS data near the jet. $Re = 3600$. Black is $\Theta < -0.0005a_o/r_o$ and white is $\Theta > 0.0005a_o/r_o$. The grey scale varies continuously between these extrema. The jet is visualized with contours of vorticity magnitude. The axial boundaries of the numerical domain (including the boundary zone) are demarked by the vertical lines. Only radiating modes are shown. Note that because of the geometry of the computational domain there is a blockage effect that decreases the sound near the axis in both the upstream and downstream directions. The sound waves appear to emanate from roughly the end of the potential core and have an apparent peak intensity at $30^\circ$ from the jet axis.

almost uniform without a strongly preferred direction. It is suggested that this low-level, almost uniform, background noise is generated by the fine-scale turbulence of the jet flow. Therefore to accurately calculate the jet noise in the peak noise direction, the large scale eddies need to be resolved.

Figure 1.3(a) shows vorticity contours downstream of the jet exit obtained by direct numerical calculations and figure 1.3(b) shows good agreement between experiment and the numerical noise calculation. For the first time, the Lighthill’s sound source was calculated directly from the flow field using Fourier methods. Though this study was performed at low Reynolds number, which does not represent the Reynolds numbers of realistic aircraft engines, it gave accurate noise data to validate the existing acoustic analogy theories and understand the turbulent
Figure 1.3. (a) Instantaneous contours of vorticity magnitude: levels are $\omega r_0/U_j = 0.35, 1, 2, 3, 4$ with lighter contours representing larger values\[18\]. (b) Far-field pressure spectrum at $\theta = 30^\circ$, --- simulation\[18\]; -- -- measurements of Stromberg et al. (See Ref. \[18\]).

Rembold et al.\[19\] evaluated the LES numerical method for a 5:1 aspect ratio rectangular jet at Mach 0.5 and compared the results with DNS calculations. The $Re$ for this calculation is 5000. The inflow is defined by a laminar top-hat velocity profile and an external disturbance is used to trigger transition. They found that the low frequency part of the far-field spectra is well reproduced by the LES using source formulations based on both the filtered velocities and the approximately deconvolved velocities. However, spurious waves from the LES data resulted in an unphysical increase of the spectral level at higher frequencies, and no subgrid-scale contribution was observed.

LES and other hybrid methods are mostly used for practical Reynolds numbers calculations. Bogey et al.\[20\] performed LES calculations in subsonic round jets at $M = 0.9$ and $Re = 65,000$ using the Smagorinsky\[14\] SGS model and calculated the noise directly from the flow field. Figure 1.4(a) shows the vorticity field obtained from the LES calculations of an $M = 0.9$ jet and figure 1.4(b) shows a comparison between the experimental and numerical Overall Sound Pressure Level (OASPL) values. This curve also shows the sound directivity with the peak around $\theta = 30^\circ$. Good agreement between the experimental and the computational values demon-
Figure 1.4. Snapshots of the vorticity field: $\omega_z$ in the $x\ y$ plane at $z = 0$.

Overall sound pressure level as a function of angle $\theta$ measured from the jet axis, at $60r_0$ from the jet nozzle. Experimental data by: + Mollo-Christensen et al.(1964); ◇ Lush (1971); × Stromberg et al.(1980).

strates the feasibility of using LES to predict accurately the turbulent flow field and the acoustic field. The LES calculations also predicted the location of the noise sources to be at the end of the potential core as suggested by several experiments. Tucker[21] used DES, Smagorinsky and monotone-integrated LES(MILES)-RANS methods to calculate unsteady flow in a plane channel. Blending of the RANS and MILES regions is achieved using a Hamilton-Jacobi type equation. The MILES-RANS approach, where the SGS method is turned off and numerical schemes are used to dissipate the downstream energy, resulted in excessive downstream decay. In these studies, the nozzle geometry was not included in the flow simulations.

Shur et al.[22, 17] performed LES calculations on complex nozzle geometries including synthetic chevrons and flight effects. The interior of the nozzle was not included in the simulations and jet conditions are specified as inflow boundary conditions at the nozzle exit. The jet noise is accurate within 2-3 $dB$ over a meaningful range of frequencies with the order of grids being 1 million points. The FW-H method was used for noise calculations with tapered funnel-shaped FW-H surfaces. Figures 1.5(a) and 1.5(b) show the effect of varying co-flow Mach number on the centerline turbulence intensity levels and in the shear layer that governs the jet noise characteristics. The turbulence intensity levels decrease with increasing co-flow.
Figure 1.5. Normalized centerline axial turbulence intensity (a) and its peak value in the shear-layer (b) with different co-flow velocities. (a): 1 - 4: CFD at $M_j = 0.9$, ratio 0.1, 0.2, 0.3, and 0.6; 5 - 8: experiments of Morris\cite{23} at $M = 0.47$, ratio 0.096, 0.206, 0.305, and 0.497. (b): 1 - 4: as in frame (a); 5, 6: analytical fits of Morris\cite{23} at high and low values of parameter $b = (U_{CL} - U_{CF})/U_j$: 0.175$b^{0.7}$ (for $b > 0.163$) and 0.3$b$ (for $b < 0.163$), respectively.

A few other important studies of subsonic and supersonic jet noise include Garnet and Estivales\cite{24}, Shih et al.\cite{25}, Choi et al.\cite{26}, Zhao, Frankel and Mongeau\cite{27}, Morris et al.\cite{28}, Anderson, Eriksson and Davidson\cite{29}, and Uzun et al.\cite{30}. These studies mainly concentrate on subsonic and supersonic round jets.

1.1.2 Dual-Stream Jets

A study of dual-stream jets is important because present day commercial aircraft, invariably, use turbofan engines. The turbofan engine is the most efficient propulsion system in operation for aircraft traveling at high subsonic cruise speeds ($M \sim 0.85$). A turbofan engine has all of the same internal components as a turbojet engine, but it is surrounded by a fan in a bypass duct. The GE90 (installed on Boeing 777 and 787 and Airbus 300) and GP7000 (installed on Airbus A380s) are some of the most powerful turbofan engines used today with high bypass ratios and dual stream exhausts. Table 1.1 lists some turbofan engines and the commercial aircrafts in service they are installed on. The ratio of the mass of air that is routed through the bypass duct to the mass of air that is routed through the core during a fixed time period is called the bypass ratio. With increased bypass ratio,
the engine becomes quieter and more efficient. The primary (core) jet exhaust is characterized by very high speeds and high temperatures, while the secondary (bypass) stream has lower speeds and is cooler. The nozzle geometry of a dual-stream jet has an important influence on the mixing and spatial evolution of the jet flow. In turn, this exerts a significant impact on the intensity and spectral distribution of the noise radiated by the jet.

The jet external flow field can be divided into three characteristic regions:

- a potential core region in which mixing is initiated in the shear layers from the nozzle lip,
- a transition region,
- a far-field or similarity region where the mixing rate is nominally proportional to the inverse of the axial distance from the nozzle.

Figure 1.6 shows the main components of the dual-stream nozzle. The initial region of a dual-stream jet consists of two shear layers: primary and secondary as shown in figure 1.6. The primary shear layer encloses the primary potential core. The region between the primary and secondary shear layers define the generalized secondary core which contains an initial potential flow region followed by a non-potential region.

The secondary jet surrounding the primary jet affects the jet noise in several ways:

- The bypass stream reduces the convective Mach number ($M_c$) of the primary jet. Equation 1.1 is used to define the convective Mach number in this

<table>
<thead>
<tr>
<th>Turbofan Engine (Bypass Ratio)</th>
<th>Commercial Aircraft Powered</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pratt &amp; Whitney JT9D (5)</td>
<td>Boeing 747, 767, Airbus A310</td>
</tr>
<tr>
<td>GE 90 (9)</td>
<td>Boeing 777</td>
</tr>
<tr>
<td>GE CF6-80A (4.66)</td>
<td>Boeing 767, Airbus A310</td>
</tr>
<tr>
<td>GE GEnx (19:2)</td>
<td>Boeing 787 and 747-8</td>
</tr>
<tr>
<td>Rolls-Royce Trent 1000 (11)</td>
<td>Boeing 787</td>
</tr>
<tr>
<td>Rolls-Royce Trent XWB (9.3)</td>
<td>Airbus A350 XWB</td>
</tr>
</tbody>
</table>

Table 1.1. High-bypass turbofan engines with the commercial aircraft on which they are installed.
Figure 1.6. Main components of dual stream jet flow: Primary potential core length, $x_p$, generalized secondary core (GSC) length, $x_{GSC}$, and protrusion of inner nozzle, $x_{prot}$ (Shupe et al.\cite{31})

The convective Mach number represents the compressibility of the shear layer. The largest amount of jet noise is believed to be generated by the large structures of the mixing layer. The convective Mach number is representative of the speed of propagation of these large instabilities. If $M_c$ is high-subsonic or supersonic, the noise production will be much greater due to the presence of Mach wave emission. Since $M_c$ is reduced between the primary jet and the surrounding flow because of the bypass stream, overall jet noise emission reduces. The velocity difference (shear) is reduced so the turbulence production is also reduced giving lower turbulence levels.

- The secondary jet lengthens the potential core of the primary jet. The majority of the large scale mixing noise comes from the end of the velocity potential core of the primary jet. Therefore, the noise sources are located further downstream in a dual-stream jet as compared to the single stream jet. After the secondary potential core and its influence disappear, the jet behaves like a single jet.

Recent hybrid methods such as the Detached Eddy Simulation (DES) have been

$$M_c = \frac{U_{fast} - U_{slow}}{a_{fast} + a_{slow}} \quad (1.1)$$
used to perform flow simulations on dual-stream nozzle geometries at subsonic and supersonic Mach numbers. Shur et al. extended the numerical methods used in Shur et al. and performed two step RANS-LES simulations on coaxial jets using grids consisting of 2-4 million nodes. The Flowcs Williams-Hawkings (FW-H) method is used for noise calculations based on tapered funnel-shaped FW-H surfaces around the jet. The SGS model is turned off downstream of the jet exit as it causes a delay in the transition to turbulence and proves too dissipative. Figure 1.7(a) shows the vorticity field downstream of a coaxial jet and figure 1.7(b) shows the OASPL directivity with still air and co-flow. The primary and secondary flow Mach numbers are $M_p = 0.75$ and $M_s = 0.85$ respectively and the co-flow Mach number is $M_{CF} = 0.28$. There is good agreement between the experimental and numerical noise spectra. Figure 1.7(b) also shows that the noise levels are lower in forward flight (co-flow) configuration because of the lower convective Mach number between the secondary jet and the surrounding flow. A similar RANS-LES numerical scheme was later used to make noise calculations for beveled coaxial jets and other complex nozzle geometries by Viswanathan et al. and good agreement between the numerical and measured values was obtained. The nozzle interior was included in the computational domain.

Several experimental studies (See Murakami et al. for other references) have been performed and empirical formulations have been proposed to study the flow and noise characteristics of subsonic and supersonic dual-stream jets. Mainly,
subsonic coaxial jets are addressed in the present Chapter. Some of the experimental studies include Murakami et al.\textsuperscript{34} where the mean flow development in coaxial nozzles, with supersonic core flow and high subsonic fan flow, is studied and a semi-empirical model is proposed for primary and secondary core lengths of coaxial jets. Viswanathan\textsuperscript{35} performed a parametric analysis of the noise generated from dual-stream jets for several core and fan flow operating conditions. In the present work, a high subsonic test case has been taken from the test matrix given in Viswanathan\textsuperscript{35} to perform dual-stream jet calculations. Khavaran and Bridges\textsuperscript{36} performed jet noise scaling of dual-stream jets using superposition methods from the spectra obtained for single stream jets.

1.1.3 Dual-Stream Jets with Installation Devices

Engine mounting devices such as pylons intrude into the bypass nozzle as shown in figure 1.8(a), thus intruding into the fan stream. The jet plume is no longer axisymmetric and hence the noise levels may vary azimuthally at a fixed observer distance from the nozzle exit. The asymmetric flow pattern also suggests that the polar variation of the noise levels may differ in and out of pylon plane. Numerical studies and experiments have been performed to model and study the effect of installation devices such as a pylon on the flow and noise characteristics of the coaxial jets. The presence of a pylon deflects the primary stream up relative to the fan stream, thickening the fan flow under the primary jet and thinning it on the top. Considering the complexity of the nozzle geometry and the escalating grid requirements in these calculations, most numerical noise studies involving a pylon have been RANS based simulations coupled with acoustic analogies.

Thomas et al.\textsuperscript{37} performed RANS simulations on five different dual-stream nozzles with chevrons and pylons to study the effect of the pylon on the jet flow characteristics. Hunter and Thomas\textsuperscript{38} performed noise calculations on jet-pylon configurations using RANS simulations and Lighthill’s acoustic analogy. Massey et al.\textsuperscript{39} performed numerical (RANS two equation $k$-$\epsilon$ model) and experimental flow analysis of dual-stream chevron and pylon configurations. Figure 1.8(b) shows the pylon configuration (Config. 6) with a coaxial nozzle and the grid lines.
Figure 1.8. (a) Twin GE CF6-50E2 turbo-fan engines mounted under the wing on a Boeing 747. (b) Structured mesh surface shown with round coaxial nozzle and pylon configuration [39].

Figure 1.9. (a) Mach number on symmetry plane of round nozzle with pylon (Config. 6). (b) Turbulent Kinetic Energy contours on symmetry plane of round nozzle with pylon (Config. 6) [39].

Figures 1.9(a) and 1.9(b) show the averaged Mach contours and TKE contours for this configuration. The numerical results are in very good agreement with the experimental data. These contours show the distortion and asymmetry in the jet plume due to the presence of pylon. Hunter et al. [40] performed RANS simulations for similar jet-pylon configurations and observed reductions in the noise levels at observer locations > 10Dj.
Eastwood and Tucker\cite{41} recently used a hybrid RANS-LES(MILES) approach using finite volume method to include the pylon in the coaxial nozzle configuration and to simulate the unsteady flow field. Recently experiments have also been performed to study the effect of a pylon on the nozzle flow. Zaman\cite{42} and Papamoschou\cite{43} studied the noise generated by nozzles including pylon and possible noise reduction by deflecting the stream in these realistic configurations. Birch et al.\cite{44} conducted experiments with a coaxial nozzle pylon configuration and concluded that the pylon introduces strong flow asymmetry in the nozzle stream. The presence of the pylon deflects the primary stream upwards relative to the secondary stream, thickening the flow at the bottom of the jet and thinning it on the top. This asymmetry is enough to give noise benefits obtained by thickening of the flow under the jet. Similarly, high lift devices such as flaps affect the jet flow in landing or takeoff configurations. These devices can be included in the computational domain to study their effect on jet flow and the noise characteristics of the jet.

Thomas et al.\cite{45} performed an experimental investigation of the jet-pylon interaction of high bypass nozzle configurations for varying tunnel Mach numbers. They observed that the pylon reduced the noise levels by 1 EPNdB compared to the baseline case and there is little effect of azimuthal angle. Shupe et al.\cite{31} conducted experiments to study the effect of a wedge-shaped fan flow deflector on the mean and turbulent flow fields of dual-stream jets. Eastwood and Tucker\cite{41} recently performed hybrid LES-RANS flow simulations of a coaxial nozzle including pylon with total number of cells ranging from 12 to 50 million. Birch et al.\cite{44} performed an experimental study of four coaxial nozzle-pylon configurations to study the effect of small asymmetries in the nozzle geometry on the noise results. They concluded that in the presence of co-flow and pylon, the fan flow deflection will not reduce the noise levels in real nozzle configurations.

Shupe and DeBonis\cite{47} performed RANS simulations of a bypass ratio 8 coaxial nozzle with an external wedge shaped noise suppressor to study the jet flow characteristics and they reported 1.1% thrust loss at takeoff conditions with the deflector. Page et al.\cite{48} conducted experimental study and performed RANS simulations of coaxial nozzles and compared the two data. The steady flow field can provide
Figure 1.10. Change in noise level due to pylon (Viswanathan and Lee\cite{46}): $M_t = 0.2$, $A_s/A_p = 3.92$, $NPR_p = 1.71$, $T_p/T_\alpha = 3.16$, $NPR_s = 1.76$, $V_s/V_p = 0.64$. $\phi = 0^\circ$, $\Delta$: $\phi = 30^\circ$, $\square$: $\phi = 60^\circ$.

inputs to a semi-empirical noise model, herein after referred to as the four source model. Recently, Viswanathan and Lee\cite{46} performed experiments in the Boeing LSAF on dual-stream jets for a range of bypass ratios with and without pylon. The forward flight effects were also assessed and they found that (1) at the lower inlet angles and up to $110^\circ$, the magnitude of the noise reduction due to forward flight is nearly uniform at all frequencies, at all velocity ratios and area ratios; (2) more complicated effects of forward flight are observed at large aft angles: $\sim 10$ $dB$ reduction at the lower frequencies and $\sim 3$ to $\sim 4$ $dB$ at the higher frequencies; (3) the area ratio plays no role at high $V_s/V_p$ but becomes important at lower $V_s/V_p$. In this study, the net nozzle exit area were kept same for the nozzles with and without pylon, so that the effect of pylon on the jet noise can be studied in isolation and the noise is not changed due to the reduction in the jet exit area in the presence of pylon. Figure 1.10 shows the difference in the levels between the axi-symmetric dual-jet nozzle and the nozzle with pylon at different azimuthal angles. In case of pylon, the sound field in the azimuthal angular range of $\phi = 0^\circ$ to $\phi = 60^\circ$ was found to be axisymmetric, except at very high engine power.
and low $A_s/A_p$ for which there was an increase in noise levels at large aft angles. The pylon is located at the azimuthal angle of $\phi = 180^\circ$. The second effect of the pylon pertains to modifications to the spectra, relative to an axi-symmetric nozzle system. A noise reduction of $\sim 2 \text{ dB}$ to $\sim 4 \text{ dB}$ was observed for the pylon nozzle at large aft angles, and an increase of $\sim 2 \text{ dB}$ at the lower inlet angles.

1.2 Noise Calculations

The most accurate method to calculate the sound field in a jet flow is to perform a direct noise calculation from the flow field. This method is computationally expensive and it takes much longer to obtain the solution. Instead, acoustic analogies can be effectively used to calculate the noise from the flow field. Even though the sound is generated by a nonlinear process, the sound field itself is known to be linear and irrotational. This implies that instead of solving the full nonlinear flow equations out to the far-field for sound propagation, one can use a computationally inexpensive method instead, such as Lighthill’s acoustic analogy (in particular, the Ffowcs Williams-Hawkings (FW-H) method) or surface integral acoustic methods such as Kirchhoff’s method both for subsonic and supersonic moving surfaces.

The governing equation of the Kirchhoff formulation for moving surfaces is an inhomogeneous wave equation in which the sources are distributed on a fictitious surface, i.e., the Kirchhoff surface, which encloses all of the physical sources. The Kirchhoff formulation is attractive because no volume integration is necessary. Unlike the FW-H source terms, however, the Kirchhoff source terms are not easily related to thickness, loading, nonlinear effects, or indeed any physical mechanisms. Another disadvantage of the Kirchhoff method is that the Kirchhoff surface must be chosen such that the inputs satisfy the wave equation. This criterion is a particular problem when enclosing the jet plume with a funnel like tapered acoustic surface as the non-isentropic and nonlinear fluid passes at the outflow disk.

Combining the mass and momentum conservation equations, Lighthill’s wave equation can be written as:

$$\frac{\partial^2 \rho}{\partial t^2} - c_s^2 \frac{\partial^2 \rho}{\partial x_j^2} = \frac{\partial^2 T_{ij}}{\partial x_i \partial x_j}$$

(1.2)
where $T_{ij} = \rho u_i u_j + \delta_{ij} \{ p - p_0 - c_0^2 (\rho - \rho_0) \} - \tau_{ij}$ is the Lighthill’s stress tensor and the viscous stress tensor $\tau_{ij}$ is given by equation \ref{eq:viscous_stress_tensor}:

$$\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right)$$ \hspace{1cm} (1.3)

The right hand side of the equation \ref{eq:equation_1.2} represents the equivalent noise sources. If the noise sources are known, the sound field can be calculated using the free space Green’s function of the wave equation. FW-H is an inhomogeneous wave equation for the external flow problem that has been embedded in unbounded space. Several integral formulations using Green’s function and generalized function theory have been proposed. The choice of the integral method depends on the application. In the present jet flow analysis a retarded time formulation is used as it requires the observer location and time to be fixed during the integration which is generally true in jet noise calculations. Numerical implementations of this formulation have proven to be very robust and efficient and most present day predictions use this formulation.

In the current work, $\tau_{ij}$ is neglected while calculating $T_{ij}$ as it is typically small because it is in the field away from surfaces\cite{3}. The second term in the tensor is also small in nearly-isentropic flow, and so is also expected to be small here. Thus for the present analysis we retain only $\rho u_i u_j$ in the source tensor. The Ffowcs Williams and Hawkings equation formulation used in the present work and the surface integration procedure are described in Section \ref{sec:2.10}.

The main disadvantage of the traditional application in the FW-H method is that to predict the noise of bodies moving at transonic speeds the quadrupole source must be included. Thus, the quadrupole, which is a volume source, ultimately requires a volume integration of the entire source region. Volume integration is computationally expensive and can be difficult to implement. Previous work\cite{22} has shown that the contribution of the quadrupole term is not significant in most subsonic applications when the integration surface is assumed to enclose all major noise sources. Therefore, this term is neglected in the present calculations.

Spalart and Shur\cite{51} studied two variants of the FW-H equation and their effect
Figure 1.11. Instantaneous vorticity and the FW-H/Kirchhoff surfaces used. $M_j = 0.77$, $TTR = 2.66$ and $M_{cf} = 0.26$. $W1$: narrowest sleeve, $L1$: shortest length of sleeve, Spalart and Shur [51].

on the jet noise simulation. The first formulation is the standard FW-H formulation written in terms of density perturbations and second formulation replaces density by a more general formulation $\rho^* = \rho_0 (1 + p'/\rho_0)^{1/\gamma}$. A difficulty with hot jets is that the local instantaneous density relative to the ambient flow, $\rho' = \rho - \rho_o$, differs significantly from the acoustic approximation $p/c_o^2$ within the jet plume. Therefore by using $\rho^*$, the FW-H surface can be placed closer to the jet plume. Figure 1.11 shows the different FW-H surfaces tested with these variants and also with Kirchhoff’s equation. Figure 1.12(a) shows the 1/3-Octave SPL calculated using Kirchhoff formulation. The $p'$-based Kirchhoff method results as shown in figure 1.12(b) are better, but the sensitivity is still unacceptable even with domain lengths of 35 diameters. The standard FW-H method is very vulnerable to the outflow disk contributions. In contrast, the variant using $\rho^*$ as shown in figure 1.12(c) is quite satisfactory, with deviations only for the shortest surface, ending at 19 diameters, which for a jet with co-flow is quite moderate.

Figure 1.13 shows an example of the noise calculation performed using Kirchhoff and FW-H surface integration from a vortex shedding cylinder [50]. In this situation it is expected that the vortices shed by the cylinder would have a very small contribution to the sound produced, hence the acoustic signal should be relatively unaffected by the placement of the integration surface. FW-H surfaces at different
Figure 1.12. Effect of surface length on the 1/3-octave spectra at $\theta = 90^\circ$ (a) Kirchhoff formulation (b) FW-H standard formulation (c) FW-H with $\rho^*$ substitution, Spalart and Shur[51].
radii are used to calculate the noise. The FW-H method predicts almost similar noise levels for all the integration surfaces (figure 1.14(b)) while Kirchhoff method does not (figure 1.14(a)). This example shows the limitation of Kirchhoff’s method when the surface is not in the linear flow region. Therefore, FW-H method is preferred over Kirchhoff’s method in jet noise calculations as it relaxes the requirement of choosing the integration surface in the linear flow region which is problem dependent.

1.3 Computational Aeroacoustics Issues

While performing the aeroacoustic calculations, several issues need to be addressed to simulate the flow-field appropriately when making reliable acoustic predictions. Some of the issues to be considered in aeroacoustic calculations are:

- Aeroacoustics problems typically involve a wide frequency range that needs to be resolved. Numerical resolution of the high frequency waves with extremely short wavelength becomes a formidable obstacle to accurate numerical simulation. Also, the length scale of the acoustic source is usually very different from the acoustic wavelength.

- In jet noise calculations, interest is in the sound waves radiated to the far field.
This requires a solution that is uniformly valid all the way from the source region to the measurement point many acoustic wavelengths away. Because of the long propagation distance, computational aeroacoustics schemes must have minimal numerical dispersion and dissipation. Also, it should propagate the waves at the correct wave speeds and should be isotropic irrespective of the orientation of the computation mesh. It should be noted that this issue can be alleviated with the use of the FW-H or Kirchhoff integral methods.

- Acoustic waves decay very slowly and actually reach the boundaries of a finite computational domain. To avoid the reflection of outgoing sound waves back into the computational domain and contamination of the solution, radiation and outflow boundary conditions must be imposed at the artificial exterior boundaries to assist the waves to exit smoothly.

- The numerical schemes used for solving the differential equations have numerical errors and a certain order of accuracy. Therefore, they tend to introduce non-physical components in the solution that need to be damped or filtered.
using an efficient artificial dissipation mechanism.

Tam\cite{52} summarized the current numerical schemes used for higher order spatial discretization, numerical dissipation and dispersion, selective artificial dissipation, temporal discretization methods such as single-step Runge-Kutta and multi-step Adams - Bashforth methods, radiation and outflow boundary conditions, tailored for aeroacoustic calculations. Many of these numerical schemes are used in the current work for efficient aeroacoustic flow field simulations.

### 1.4 Objectives of the Current Work

The main objective of the current work is to make reliable jet noise predictions for high-subsonic single and dual-stream jets, and to achieve predictions with acceptable engineering accuracy using moderate computational resources. The modified DES approach with numerical methods tailored for aeroacoustic applications is used in the current work to simulate the jet flow. The FW-H surface integration method is used to calculate the far-field noise at different observer locations. The effect of co-flow and the jet noise scaling is studied and analyzed at aircraft takeoff conditions. The two-point turbulence statistics and length and time scales, for single and dual-stream jets are analyzed at varying co-flow Mach numbers. The effect of an engine installation device such as pylon on the jet noise is also studied. The cross-sectional area of the nozzle with pylon is altered to match the area ratio of the baseline dual-stream nozzle.

### 1.5 Original Contributions of the Current Work

The present solver EAGLEJet has been developed from the non-linear disturbance equations solver previously developed for wind-turbine blade noise\cite{53} applications. The solver is extended to work with the multi-block structured grid with arbitrary orientations which is required to perform simulations on complex nozzle geometries. The data communication strategy has been implemented for two types
of block interface conditions, one where the grid points at two block boundaries match with each other and the second, where the grid points at two blocks do not match. The non-matching boundaries are used to refine grid near the pylon walls in the nozzle with pylon configuration. Several convergence acceleration methods such as dual-time stepping method and implicit residual smoothing method are implemented and optimized for the present nozzle geometries. Several artificial dissipation switches have been implemented and assessed for the nozzle cases as the external dissipation will affect the onset of turbulence and development of shear layer instability in the jet flows. A modified DES approach is used to avoid excessive damping of the small-scale eddies in the jet plume region. An efficient parallel communication method for structured multi-block grid is implemented where the data are efficiently divided among the processors and the communication overhead and memory requirements are minimized by storing the minimum required data at the block interfaces.

Unlike the majority of previous work, which have excluded the nozzle geometries from the computations and assumed unrealistic inlet conditions at the nozzle exit, complex nozzle geometries are included in the present computations. Without using any artificial excitation, a finite nozzle thickness is used to trigger the unsteady flow in the jet plume. The effects of grid refinement, selective artificial dissipation terms, and the size and location of the acoustic data surface on the resolution of the jet noise simulations are studied to provide guidelines for jet noise predictions on complex nozzle geometries.

The flight effects are quantified by performing simulations for single stream jets with varying co-flow Mach number. The moving source term is included in the Ffowcs Williams and Hawkings equation formulation to accurately incorporate the effect of parallel stream in far-field noise calculation. Most of the previous work to study and model the flight effects have been experimental and discrepancies have been observed in the data obtained at different experimental facilities\cite{54, 55, 56}. The flight effects assessed in the current work gives an insight about the discrepancies observed in the experiments.

Most previous numerical studies on dual-stream jets performed RANS simulations
and used acoustic models for noise predictions. In the present work, the modified DES approach is used to obtain the unsteady time accurate flow and the noise is calculated from the time accurate acoustic data sampling. In the previous works, the flight effects were only analyzed for the single stream jets. Since most commercial aircraft engines have turbo-fans with a dual-exhaust, it becomes necessary to study the flight effects at takeoff configurations for dual-stream nozzles. A method to perform the analysis of the flight effects for the dual-stream jets is proposed and the flight effects are quantified for different jet operating conditions. A comparison has been made between the flight effects observed for single and dual-stream jets. Statistical analysis of jet flow has been previously performed for single stream jets without parallel stream. In the present work, a detailed statistical analysis has been performed for both single and dual-stream jets with varying parallel stream velocities.

The engine installation effects have been studied by including a simplified model of the actual pylon structure with the dual-stream nozzle. Most of the previous numerical studies have used steady state RANS simulations to get the flow field. The unsteady flow simulation increases the complexity of the computational grid and the numerical simulation. Since pylon reduces the actual nozzle exit area of the fan-nozzle, the fan-nozzle with pylon is modified to keep the nozzle exit area ratio the same as the nozzle without the pylon. This separates the effect on the noise due to change in the nozzle area ratio from the effect due to the pylon. This is a novel approach and has not been used in a numerical study before.

1.6 Outline of the Thesis

Chapter 2 explains the numerical approaches used for flow and noise simulation in the present study. Chapter 3 lists some of the validation and benchmark cases used for the flow and noise solvers. Chapter 4 presents the flow and noise simulation results for the single stream jets operating at $M_j = 0.9$ with varying total temperature ratios and co-flow Mach numbers. Chapter 5 presents the flow and noise simulation results for the dual-stream jets operating at high subsonic core and fan
stream Mach numbers for both unheated and heated core flows. The forward flight
effects are presented for both single and dual-stream jets. Chapter 5 also presents
the flow and noise simulation results for dual-stream nozzle with a simplified pylon
structures. The results are compared with the baseline dual-stream nozzle. Chapter 6 lists the conclusions made from this study followed by suggestions for future
work.
Chapter 2

Numerical Methods

The main objective of the current research is the noise prediction of high subsonic single and dual stream jets. The effect of the installation devices such as a pylon on jet noise generation is also investigated. The DES approach is used to simulate the turbulent flow field. Far field acoustic calculations are performed by using the near-field unsteady data sampled on a FW-H\[29\] penetrable acoustic data surface. A Navier-Stokes solver named EAGLEJet has been developed to perform the flow calculations. The Spalart-Allmaras RANS turbulence model \[57\] is used to model the turbulence scales. In the hybrid RANS/LES approach, the eddy viscosity is turned off ($\mu_t = 0$) outside the jet exit and the grid spatial resolution is used to filter the unresolved turbulence scales. A sub-grid scale (SGS) model (e.g. Smagorinsky type model) is generally used in LES implementation to model the sub-grid scales. In the present solution, no SGS model is used as it have been found to be too dissipative in the free jet shear layer region thus suppressing the required turbulence levels. The turbulence characteristics of the jet affect the far-field noise predictions. The present DES approach is similar to the Monotonically Integrated LES (MILES)\[21\] method where the inherent dissipation of the numerical scheme is used to dissipate the smaller scales.

The main features of EAGLEJet include: a one equation Spalart-Allmaras turbulence model; selective artificial dissipation\[58, 59, 60\]; low-dissipation and low-dispersion Runge-Kutta (LDDRK)\[61\] explicit time integration schemes; dual time-
stepping\,[58]\); dispersion relation preserving (DRP)\,[62]\ high order finite difference scheme; and, parallel domain decomposition with multi-block approach. The characteristic interface conditions\,[63]\,[64]\ are used for the data communication between the grid blocks. Artificial excitation is not used to trigger the turbulence in the jet flow. Instead, the finite thickness of the nozzle lip is used to trigger an absolute instability and drive the shear layer unsteadiness.

The following sections describe the numerical approaches used in the code followed by a history of the code development.

2.1 Hybrid RANS/LES Turbulence Modeling

As mentioned before, both large scales and fine scales of turbulence are responsible for the broadband noise generation in the jet flows. Therefore, turbulence modeling is of crucial importance to jet noise simulation. DNS has the merit of resolving every detail of the turbulent structures without applying any turbulence model. Limited computer resources, however, prevent its application in practical jet noise problems. Fortunately, for the jet noise calculations we do not need to know everything about turbulence\,[65]\ . If the most energetic part of the turbulent spectrum is resolved, a satisfactory noise prediction can be achieved. The large scales carry most of the energy locked in them which governs the noise characteristics of the jet. Therefore, the emphasis is on capturing the large scales of turbulence and to efficiently model or filter the smaller scales.

The one equation Spalart-Allmaras turbulence model is used to implement the RANS part of the DES approach as it is computationally inexpensive and very robust. The instantaneous Navier-Stokes equations in tensor form can be written as:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \tag{2.1}
\]

\[
\frac{\partial \rho u_j}{\partial t} + \frac{\partial \rho u_j u_i}{\partial x_i} = -\frac{\partial p \delta_{ij}}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} \tag{2.2}
\]

\[
\frac{\partial \rho e}{\partial t} + \frac{\partial (\rho e + p) u_i}{\partial x_i} = -\frac{\partial q_i}{\partial x_i} + \frac{\partial u_j \tau_{ij}}{\partial x_i} \tag{2.3}
\]
To implement them in computational domain, these equations are re-written in Cartesian coordinates as:

$$\frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = \frac{\partial E_V}{\partial x} + \frac{\partial F_V}{\partial y} + \frac{\partial G_V}{\partial z}$$

(2.4)

The terms on the left hand side are the inviscid terms given in equations B.18-B.22. Right hand side terms represent the viscous terms. At the Reynolds numbers typically present in realistic jet cases, these equations have very chaotic turbulent solutions, and it is necessary to model the influence of the smallest scales or eddies. Most turbulence models are based on one-point averaging of the instantaneous equations. In the compressible flow equations, it is convenient to use density weighted time average decomposition called Favre averaging to avoid the introduction of turbulence stress terms in the mass conservation equation. In most cases it is not necessary to use Favre averaging though, since turbulent fluctuations most often do not lead to any significant fluctuations in density. In that case, more simple Reynolds averaging can be used. Only in highly compressible flows and hypersonic flows is it necessary to perform the more complex Favre averaging. Favre averaging is performed on the flow variables in this present work where a significantly high density gradient is present in the heated subsonic jets.

A instantaneous value \( \Phi \) can be decomposed into the time averaged value and the fluctuating part \( \Phi' \) as:

$$\Phi = \bar{\Phi} + \Phi'$$

(2.5)

The classical time dependent or Reynolds averaged value \( \bar{\Phi} \) can be calculated as:

$$\bar{\Phi} = \frac{1}{\Delta t} \int_{t}^{t+\Delta t} \Phi dt$$

(2.6)

where \( \Delta t \) is the averaging time. A density weighted time averaging is defined in equation 2.7 as:

$$\bar{\Phi} = \bar{\rho} \bar{\Phi} / \bar{\rho}$$

(2.7)

where an over-bar represents a Reynolds averaged variable and a tilde represents
a Favre averaged variable. The Favre averaged value is a mass averaged value. It should be noted that in the present definitions, $\overline{\Phi'} = 0$ but $\overline{\Phi''} \neq 0$. The variable can now be decomposed into mass averaged flow part and the fluctuating part. For example, the instantaneous velocity $u$ can be decomposed as:

$$u = \bar{u} + u''$$  \hspace{1cm} (2.8)

where $\bar{u}$ is the Favre averaged velocity component which can be calculated from equation 2.9 as:

$$\bar{u} = \frac{1}{\bar{\rho} \Delta t} \int_{t}^{t+\Delta t} \rho u dt$$  \hspace{1cm} (2.9)

The Favre averaged continuity and momentum equations can be calculated in a similar manner and are given as equations 2.10 and 2.11:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial (\bar{\rho} \bar{u}_i)}{\partial x_i} = 0$$  \hspace{1cm} (2.10)

$$\frac{\partial (\bar{\rho} \bar{u}_i)}{\partial t} + \frac{\partial (\bar{\rho} \bar{u}_j u_i)}{\partial x_j} = -\frac{\partial (\bar{\rho} \delta_{ij})}{\partial x_i} + \frac{\partial (\bar{\tau}_{ij} + \bar{\rho} \Gamma_{ij})}{\partial x_j}$$  \hspace{1cm} (2.11)

Equation 2.11 shows the averaged momentum equation in tensor notation. $u_i$ represents the velocity component in $i^{th}$ direction, $\rho$ is the density, $p$ is the pressure, $\tau$ is the viscous stress tensor, $x_i$ is the Cartesian coordinate in the $i^{th}$ direction and $t$ is the time. The last term in the right hand side of equation 2.11 is the turbulence stress term given as:

$$\bar{\rho} \Gamma_{ij} = -\rho u''_j u''_i = \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \mu_B \frac{\partial u_k}{\partial x_k} \delta_{ij} \right]$$  \hspace{1cm} (2.12)

The nonlinear term $\bar{\rho} \Gamma_{ij}$ is approximated by the Boussinesq eddy viscosity approximation which says that the stress tensor can be modeled with an eddy viscosity and trace-less mean strain rate tensor as shown in equation 2.12. $\mu$ is the molecular viscosity, $\mu_B$ is the Bulk or second viscosity and $\delta_{ij}$ is the Kronecker delta.
hypothesis can be used to relate the molecular viscosity to the Bulk viscosity as:

$$\mu_B = -\frac{2}{3}\mu$$ (2.13)

This assumption is valid in the present simulations as the smallest wavelength of the acoustic waves of interest is much larger than the mean free path of the air molecules. Equation 2.14 lists the Favre averaged energy equation:

$$\frac{\partial (\bar{\rho}\tilde{E}_t)}{\partial t} + \frac{\partial \left[ (\bar{\rho}\tilde{E}_t + \bar{p})\tilde{u}_j \right]}{\partial x_j} = -\frac{\partial \left[ q_{L,j} + q_{T,j} \right]}{\partial x_j}$$

$$-\frac{\partial \left[ \tilde{u}_i (\tilde{\tau}_{ij} + \bar{p}\Gamma_{ij}) \right]}{\partial x_j}$$

$$+ \frac{\partial \left[ u''_i \tilde{\tau}_{ij} - \frac{1}{2} \rho u''_j u''_i u''_i \right]}{\partial x_j}$$ (2.14)

The first term on the left hand side of equation 2.14 can be calculated as:

$$\bar{\rho}\tilde{E}_t = \bar{\rho}(\bar{e} + \tilde{u}_i\tilde{u}_i/2) + \bar{\rho}u''_j u''_i / 2$$ (2.15)

where $E_t$ is the total energy and $\bar{e}$ is the specific internal energy. $q_j$ on the right hand side of equation 2.14 is the heat flux where the subscripts $L$ and $T$ denote the laminar and turbulent heat flux components respectively. The laminar heat flux is calculated using the Fourier law:

$$\overrightarrow{q_{L,j}} = -k \nabla T$$ (2.16)

where $k$ is the thermal conductivity which is treated as constant for air and is calculated as $k = C_p\mu/Pr$. The turbulent heat flux can be calculated as:

$$q_{T,j} = \bar{\rho}u''_j u''_i = -\frac{\mu_t}{Pr_T} \frac{\partial T}{\partial x_j}$$ (2.17)

using the gradient diffusion hypothesis. $\mu_t$ is the turbulent eddy viscosity calculated from the eddy viscosity equation. The turbulent Prandtl number $Pr_T$ is
assumed to have a constant value of 0.9. The assumption of continuum flow is valid for the current flow conditions. The last term on the right hand side of the energy equation, which represents molecular diffusion and turbulent transport of turbulent kinetic energy, is ignored as its contribution is significant only in the hypersonic regime.

The state equation of a perfect gas $\bar{p} = \bar{\rho}RT$ is used to relate pressure, temperature and density. The molecular viscosity, $\mu$, is calculated using the Sutherland’s law:

$$\mu = C_1T^{3/2}/(T + C_2) \quad (2.18)$$

where $C_1 = 1.456 \times 10^6 \text{kg}/(\text{ms}\sqrt{K})$ and $C_2 = 110.4 \text{K}$ for air. The bulk or second viscosity $\mu_t$ is obtained by solving an eddy-viscosity equation (2.19) which is written in terms of an eddy-viscosity like variable $\hat{\nu}$.

$$\frac{\partial \hat{\nu}}{\partial t} + \bar{u}_j \frac{\partial \hat{\nu}}{\partial x_j} = c_{b1} \hat{S}\hat{\nu} - c_{v1}\bar{f}_w \left(\frac{\hat{\nu}}{d}\right)^2 + \frac{1}{\sigma} \frac{\partial}{\partial x_k} \left[(\nu + \hat{\nu}) \frac{\partial \hat{\nu}}{\partial x_k}\right] + \frac{c_{b2}}{\sigma} \frac{\partial \hat{\nu}}{\partial x_k} \frac{\partial \hat{\nu}}{\partial x_j} \quad (2.19)$$

The first term on the right hand side of equation (2.19) is the production term, second term is the wall destruction term which is used in the DES implementation and the third and fourth terms are the turbulent diffusion terms. $\nu$ is the kinematic viscosity and $d$ is the distance to the nearest wall in equation (2.19). The turbulent eddy viscosity $\mu_t$ can now be calculated from equation (2.20) as:

$$\mu_t = \bar{\rho}\hat{\nu}f_{\nu1} \quad (2.20)$$

The model constants and empirical terms in the transport equation (2.19) are given as:

$$c_{b1} = 0.1355 \quad (2.21)$$
$$c_{b2} = 0.622 \quad (2.22)$$
$$c_{v1} = 7.1 \quad (2.23)$$
$$\sigma = 2/3 \quad (2.24)$$
The governing equations presented above are solved in generalized coordinates using high order spatial discretization and time integration methods. The equations in generalized coordinates can be found in detail in Appendix A.

The traditional DES formulation has been formulated for the Spalart-Allmaras model and is implemented by solving the eddy-viscosity equation and by replacing the distance $d$ in the destruction term by equation (2.38):

$$\tilde{d} = \min(d, C_{DES}\Delta)$$

where $\Delta$ represents the largest grid spacing of the mesh cell in three directions in the space $\Delta = \max(\Delta x, \Delta y, \Delta z)$, and $C_{DES}$ is the DES coefficient set in homogeneous turbulence to be 0.65. So Spalart-Allmaras model based DES acts as LES with a wall model. A reasonable RANS solution can be obtained by
setting $\tilde{d} = d$ in the computational domain. In the present work, a modified form of the DES method is used, which is tailored for jet simulations. The traditional DES model was found to be too dissipative downstream of the nozzle exit and suppressed the turbulence levels required for accurate noise calculations. To implement the DES model, $\mu$ is replaced by $\mu_{\text{tot}}$ in equation (2.12).

$$\mu_{\text{tot}} = \mu + \mu_t$$  \hspace{1cm} (2.39)

In the modified DES approach, the eddy viscosity $\mu_t$ is set to zero in the DES region in equation (2.39) and the grid spacing is used to filter the smaller scales.

### 2.2 Dispersion Relation Preserving (DRP) scheme

In computational acoustics, the acoustic waves are computed directly from the governing equations of the compressible flows, namely, the Euler equations or the Navier-Stokes equations. Invariably, the discretized equations behave mathematically like a dispersive wave system even though the waves supported by the original partial differential equations are non-dispersive. For accurate noise calculations it is important that the finite difference scheme is less dispersive in nature. The Dispersion Relation Preserving (DRP) schemes introduced by Tam and Webb are used in the current research because they are optimized to reduce wave dispersion. The formulation of these schemes and the derivation of the coefficients are briefly discussed here.

The dispersion characteristics of general finite difference schemes can be illustrated using the one-dimensional convective wave equation (2.40):

$$\frac{\partial u}{\partial t} + c \frac{\partial u}{\partial x} = 0$$  \hspace{1cm} (2.40)

with the initial condition $u(x,0) = \varphi(x)$. $\omega = c\alpha$ is the dispersion relation of the
wave equation. The group velocity or the wave speed is given as \( c = d\omega/d\alpha \). All wave components propagate with the same speed: hence, they are non-dispersive. The finite difference approximation of the spatial derivative of \( u \) with a stencil of \( 2N + 1 \) points can be written as:

\[
\left( \frac{\partial \bar{u}}{\partial x} \right)_l \approx \frac{1}{\Delta x} \sum_{j=-N}^{N} a_j u_{l+j}
\]

(2.41)

For example, \( N = 3 \) for seven point stencil. Taking the Fourier transform of both sides of equation 2.41 yields:

\[
i\bar{\alpha} \hat{u} \cong \frac{1}{\Delta x} \sum_{j=-N}^{N} a_j e^{ija\Delta x} = \frac{2i}{\Delta x} \hat{u} \sum_{j=1}^{N} a_j \sin(\alpha \Delta x)
\]

(2.42)

where \( \alpha \) is the wavenumber of the continuous variable \( u \), whereas the effective wavenumber \( \bar{\alpha} \) appears in the Fourier transform of the discretized equation. For a central difference scheme, \( \bar{\alpha} \) is real whenever \( \alpha \) is real. \( \hat{u} \) is the Fourier transform of \( u \). Coefficients of standard central difference schemes in equation 2.41 are derived by means of Taylor series expansions. These schemes may not have good dispersion preserving characteristics. By lowering the order of accuracy of discretization, one free coefficient can be assigned. This coefficient can then be determined by minimizing the square of the difference between \( \alpha \Delta x \) and \( \bar{\alpha} \Delta x \) (Equation 2.43) over the desired range of wave numbers.

\[
E = \int_{0}^{\beta} |\bar{\alpha} \Delta x - \alpha \Delta x|^2 d(\alpha \Delta x)
\]

(2.43)

Other coefficients can then be calculated using the free parameter. Figure 2.1 shows the dependence of \( \bar{\alpha} \Delta x \) on \( \alpha \Delta x \) over the interval \( -\pi \leq \alpha \Delta x \leq \pi \) for 6th order central difference, seven point DRP and 15 point DRP schemes. For low wave numbers \( \bar{\alpha} \Delta x \) is nearly equal to \( \alpha \Delta x \). It can be seen from the figure 2.1 that for higher wave numbers, the DRP schemes have a smaller error than the Taylor series based methods. As compared to the central difference scheme, there is some improvement in the error in 7-point DRP scheme and there is no additional
In the present work, a seven point DRP stencil is used for spatial discretization as it sufficiently optimizes the dispersive properties of the numerical scheme. The coefficients for the seven point central difference scheme are obtained as shown in equation (2.44):

\[ a_0 = 0 \]
\[ a_{-1} = -a_1 = -0.763289242 \]
\[ a_{-2} = -a_2 = 0.160631393 \]
\[ a_{-3} = -a_3 = -0.019324515 \]

Similarly, coefficients for the biased operators near the boundaries are used as given in Tam[66]. In the present work, either DRP biased operators or lower order stencils are used near the boundary points.
2.3 Time-Marching Schemes

The following section describes the time marching schemes used in the present work. Implementation procedure for the following time marching schemes is described: low-dispersion low-dissipation Runge-Kutta explicit time marching, standard fourth order Runge-Kutta time marching, local time marching and dual-time stepping algorithms.

2.3.1 Low-Dissipation Low-Dispersion Runge-Kutta Scheme

A low dissipation and low dispersion explicit time integration scheme suited to aeroacoustic applications is implemented in the present solver. These schemes are based on explicit Runge-Kutta schemes and were introduced by Hu et al. [61] in 1996. Runge-Kutta schemes are multistage methods and the coefficients of the Runge-Kutta schemes are chosen such that the maximum possible order of accuracy is obtained for a given number of stages. These coefficients can also be chosen such that the dissipation and dispersion errors for the propagating waves are minimized. These optimized schemes are referred to as low-dissipation and low-dispersion Runge-Kutta (LDDRK) schemes. The Fourier analysis of the convective wave equation as presented in the previous section can be used to determine the optimized coefficients for the time integration scheme. The error equation for a p-stage Runge-Kutta scheme in this case is given by equation 2.45 as:

\[
E_i = \int_0^1 \left| 1 + \sum_{j=1}^{p} c_j (-i\mu)^j - e^{-i\mu} \right|^2 d(\mu)
\] (2.45)

where \( \mu = c\alpha \Delta t \) and the \( c_j \) are calculated by minimizing the error equation. A general p-stage Runge-Kutta scheme can be written as \( K_i = \Delta t F(u^n + \tilde{\beta}_i K_{i-1}) \) where \( u^n \) is the vector containing solution values at spatial mesh points, \( \tilde{\beta}_i \) are
<table>
<thead>
<tr>
<th>Stages</th>
<th>$c_1$</th>
<th>$c_2$</th>
<th>$c_3$</th>
<th>$c_4$</th>
<th>$c_5$</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>1</td>
<td>0.5</td>
<td>0.166558</td>
<td>0.0395041</td>
<td>0.00781071</td>
</tr>
</tbody>
</table>

Table 2.1. Coefficients for five stage LDDRK scheme [61]

Constants with $\bar{\beta}_1 = 0$ and are related to $c_j$ by,

\[ c_2 = \bar{\beta}_p \]
\[ c_3 = \bar{\beta}_p \bar{\beta}_{p-1} \]

....

\[ c_p = \bar{\beta}_p \bar{\beta}_{p-1} \cdots \bar{\beta}_2 \]

The coefficients for the five stage LDDRK scheme are given in table 2.1. The five stage LDDRK scheme is implemented in the current solver and has a higher stability limit as compared to the 4 and 6 stage schemes.

2.3.2 Traditional Runge-Kutta Methods

The Runge-Kutta time integration methods, developed around 1900 by the German mathematicians C. Runge and M. W. Kutta, are a family of implicit and explicit iterative methods to solve ordinary differential equations. They are non-linear with a high order of accuracy and are limited to two time levels. A general K-stage Runge-Kutta method can be written as:

\[ U^{(1)} = U^n \]
\[ U^{(2)} = U^n + \Delta t \alpha_2 H^{(1)} \]
\[ U^{(3)} = U^n + \Delta t \alpha_3 H^{(2)} \]

....

\[ U^{(K)} = U^n + \Delta t \alpha_K H^{(K-1)} \]
\[ U^{n+1} = U^n + \Delta t \sum_{k=1}^{K} \beta_k H^{(k)} \]
and for consistency equation (2.48) should be satisfied.

\[ \sum_{k=1}^{K} \beta_k = 1 \quad (2.48) \]

The notation \( H^{(k)} \) implies \( H^{(k)} = H(U^{(k)}) \) for the case where \( H \) is independent of time which is the case in the hyperbolic Navier-Stokes governing equations. The classic fourth order Runge-Kutta method for time accurate marching is defined by the following coefficients:

\[
\begin{align*}
\alpha_2 &= \frac{1}{2} \\
\alpha_3 &= \frac{1}{2} \\
\alpha_4 &= 1 \\
\beta_1 &= \frac{1}{6} \\
\beta_2 &= \beta_3 = \frac{1}{3} \\
\beta_4 &= \frac{1}{6}
\end{align*}
\quad (2.49)
\]

Note that for each number of the stages \( K \), there is an infinite number of possible Runge-Kutta schemes with maximum order of accuracy. For steady analysis, it is important to allow the greatest possible time step, and therefore the extension of the stability region is more important than their order. Thus, the order of Runge-Kutta method with particular coefficients can be relaxed to obtain a higher stability region hence, and larger time steps can be used to speed up the convergence to steady state solution. The modified coefficients \( \alpha \) for steady state calculations are listed below:

\[
\begin{align*}
\alpha_1 &= 0.375 \\
\alpha_2 &= 0.5 \\
\alpha_3 &= \alpha_4 = 1.0
\end{align*}
\quad (2.50)
\]

The traditional four step Runge-Kutta (RK4) time marching scheme is used for unsteady time accurate flow simulations and the steady state calculations are performed using the coefficients given in equation (2.50). In the dual-time stepping method, steady state sub-iterations are performed in fictitious time while time accurate marching is used in the physical time.
2.3.3 Local Time Stepping

Local time stepping is used in steady state viscous calculations, where cell size is not uniform in the grid. This allows use of the maximum local time step, without violating locally the numerical stability, and also accelerates convergence. Local time stepping is used in conjunction with the RK4 steady time marching scheme described in the previous section. The local time step is calculated by picking the minimum value from the inviscid and viscous time step values as:

$$\Delta t = \min(\Delta t_I, \Delta t_V)$$

(2.51)

The inviscid time step is calculated using the Courant Friedrichs Lewy (CFL) condition criterion as given in equation 2.52:

$$\Delta t_I = \frac{\text{CFL}}{\lambda_\xi + \lambda_\eta + \lambda_\zeta}$$

(2.52)

where $\lambda_\xi$, $\lambda_\eta$ and $\lambda_\zeta$ are eigenvalues of the Jacobian of the coordinate transformation matrix. The viscous time step ($\Delta t_V$) is based on the local Reynolds number ($Re$), CFL number and Mach number $M$.

$$\Delta t_V = \frac{\rho Re CFL}{M \mu (|\xi| + |\eta| + |\zeta|)}$$

(2.53)

where $|\xi| = \sqrt{\xi_x^2 + \xi_y^2 + \xi_z^2}$. $|\eta|$ and $|\zeta|$ are calculated in a similar manner.

2.3.4 Dual Time Stepping

The dual time stepping method is used to accelerate the convergence and to relax the time step restriction normally implemented for stability considerations. This approach was first introduced by Jameson[67] where he implemented this method for flow calculations past airfoils and wings. The basic idea is to introduce a fictitious time step into the computation, so that the unsteady problem becomes a
pseudo-steady problem. This brings in inherent advantages associated with steady-state problems like local time stepping and implicit residual smoothing. With the introduction of a fictitious time step, the discretized Navier-Stokes equations can be written as equation (2.54)

\[
\frac{\partial Q}{\partial \tau} = -\frac{\partial Q}{\partial t} + \text{Res}(Q) = \text{Res}^*(Q) \quad (2.54)
\]

The physical time step \(\Delta \tau\) is generally fixed due to some constraints, such as the maximum Strouhal number resolution required, while the pseudo-time step \(\Delta t\) is calculated from the steady-state problem. With an efficient steady-state solver, it is possible to drive the modified governing equation to a steady state in the fictitious time. On convergence in fictitious time the modified residual term goes to zero i.e., \(\text{Res}^*(Q) \approx 0\) and the original equations are recovered. This approach removes the stability constraint on the value of physical time step and thus allows the taking of large physical time steps and the ability to reach a converged solution faster.

The physical time derivative at the next time step is calculated using a three-point backward discretized scheme as:

\[
\left[ \frac{\partial Q}{\partial \tau} \right]^{n,m+1} = -\frac{Q^{n,m-2} - 4Q^{n,m-1} + 3Q^{n,m}}{2\Delta t} + \text{Res}(Q^{n,m}) = \text{Res}^*(Q^{n,m}) \quad (2.55)
\]

where, \(m\) represents the physical time step and \(n\) represents the fictitious time step. Between each physical time step, the flow solution is marched in fictitious time step using a four stage Runge-Kutta method and the number of sub-iterations can be varied according to the problem. The criterion is that the solution should be converged well in fictitious time step before marching onto the next physical time step. When the flow solution is converged well, the flow reaches a quasi-steady state represented by, \(\text{Res}^*(Q) \approx 0\) and \(Q^{n+1,m+1} = Q^{n,m+1}\). Convergence acceleration methods are used in the fictitious time marching to improve the computational efficiency.
2.4 Artificial Dissipation

Compact difference discretization, like other centered schemes, is non-dissipative and is therefore susceptible to numerical instabilities due to the growth of high-frequency modes. These difficulties originate mainly from mesh non-uniformity, boundary conditions and nonlinear flow features. In order to increase the stability limits, a high-order implicit filtering technique\cite{68} and a high order adaptive dissipation scheme\cite{58} are incorporated in the present solver. Their formulations are discussed in the subsequent sub-sections.

2.4.1 Sixth Order Low Pass Filter

A sixth order accurate filtering scheme developed by Visbal and Gaitonde\cite{68} is implemented to dissipate the spurious high frequency content in the spatial discretization schemes. If a component of the solution vector is denoted by $\phi$, filtered values $\hat{\phi}$ at the inner points are obtained by solving the tridiagonal system:

$$\alpha_f \hat{\phi}_{i-1} + \hat{\phi}_i + \alpha_f \hat{\phi}_{i+1} = \sum_{n=0}^{3} \frac{a_n}{2} (\phi_{i+n} + \phi_{i-n})$$

(2.56)

where $a_n$ are the constants derived using the Taylor series and Fourier series analysis. $n$ goes from 0 to 3 as a sixth order filter is desired. This tridiagonal system can be solved using the traditional Thomas algorithm. The points near the boundary are filtered using the equations 2.57 and 2.58.

$$\alpha_f \hat{\phi}_{i-1} + \hat{\phi}_i + \alpha_f \hat{\phi}_{i+1} = \sum_{n=1}^{7} a_{n,i} \phi_n$$

(2.57)

$$\alpha_f \hat{\phi}_{i-1} + \hat{\phi}_i + \alpha_f \hat{\phi}_{i+1} = \sum_{n=0}^{6} a_{N-n} \phi_{N-n}$$

(2.58)

$N$ is the total number of grid points in a given direction. The coefficients $a_n$ for a sixth order filter for the internal points are given in table 2.2. The coefficients
used for the points near the boundary can be found in Ref. [69]. No dissipation is applied at the boundary. The adjustable parameter $\alpha_f$ satisfies the inequality $-0.5 \leq \alpha_f \leq 0.5$ with higher values of $\alpha_f$ corresponding to a less dissipative filter. Extensive numerical experience [69] suggests that regardless of the time-integration scheme, values of $\alpha_f$ between 0.3 and 0.5 are appropriate. The filter is typically chosen to be at least two orders of accuracy higher than the difference scheme. Since a fourth order accurate DRP scheme is used in the present research, the sixth order filter is chosen. The filter is applied to the conserved variables sequentially in each coordinate direction. Although the frequency of application can be varied, normally the solution is filtered once after the final stage of the explicit LDDRK method. In jet flow simulations, one needs to be careful about over-damping the waves required for accurate noise calculations. Since this filter applies an uniform damping in the whole domain, a selective artificial dissipation mechanism described in the next section is used for unsteady jet flow calculations.

### 2.4.2 Selective Artificial Dissipation

A higher order version of the adaptive artificial dissipation developed by Jameson [67] is implemented in the present solver. An optimized smoother using a seven point stencil is used as background dissipation. Standard second and fourth derivative dissipation is also employed near the boundaries and discontinuities. A switch $\mu$ is used to decide the dissipation value. The cutoff is based on the maximum of the fourth derivatives of pressure and density. A scaling coefficient $\kappa$ is set to 0.35. The maximum value of the switch is set to $\mu_{\text{max}} = 0.12$. The switch is calculated as:

$$\mu = \min \left( \mu_{\text{max}}, \max \left( \kappa \frac{1}{\rho} \frac{d^4 \rho}{d \xi^4}, \kappa \frac{1}{p} \frac{d^4 p}{d \xi^4} \right) \right) \tag{2.59}$$

A combination of a pressure-based switch, a pressure and density-based switch and

<table>
<thead>
<tr>
<th>$a_0$</th>
<th>$a_1$</th>
<th>$a_2$</th>
<th>$a_3$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\frac{11}{16}$ + $\frac{3\alpha_f}{8}$</td>
<td>$\frac{15}{32}$ + $\frac{11\alpha_f}{16}$</td>
<td>$-\frac{3}{16}$ + $\frac{3\alpha_f}{8}$</td>
<td>$\frac{1}{32} - \frac{\alpha_f}{16}$</td>
</tr>
</tbody>
</table>

**Table 2.2.** Coefficients for 6th order low pass filter, $-0.5 \leq \alpha_f \leq 0.5$
an only density-based switch can be used. At boundaries, the dissipation stencil is reduced from seven points to five points and then to three points. This scheme is explained in more detail by Lockard[58]. The artificial dissipation is added to the residual after the first sub-step of the explicit time integration scheme.

2.5 Immersed Boundary Method

This method is implemented in the current work to incorporate small geometric changes in the body without changing the body fitted grid. Body fitted structured meshes are hard to create for a small geometric body in complex nozzle shapes especially including installation devices such as pylon. To overcome this problem, an immersed boundary method approach is used where the governing equations are modified at few grid points to replicate the solid body as required. Numerical tests [70] have shown that the effects of small geometric features in the case of jet flows can be modeled satisfactorily without requiring a full-body conformal mesh over all fine geometric details.

An Immersed Boundary Method (IBM) approach can also be used to model the installation devices which have small geometric bodies but in the present work, body-fitted grid is used to model the simplified nozzle-pylon configuration. An IBM does not require a body-conformal mesh on the solid surfaces. The solid body is modeled within the mesh such that few grid points are embedded inside the body and the others lie outside. The actual solid wall boundary is replaced by the grid points that lie nearest to the solid wall boundary. Therefore, the requirement for better representation of the solid body is that the grid points should be clustered near the solid wall boundary.

There are two parts to the implementation of the IBM. The first task includes marking the grid points in the computational domain that lie inside the solid body efficiently. The Line Intersection Method (LIM)[71], a preprocessing code is used to mark the grid points that lie inside the solid body. Figure 2.2 shows the idea of the LIM method. If a line is drawn along the grid points from one end to the other end of the domain, it will either intersect the body an even number of times
Figure 2.2. Actual geometry boundary (blue) and computational marked boundary (green) used in immersed boundary method. Three points (black) show the three possibilities for a grid point.

or an odd number. The first case depicts that the point lies outside the solid body while the second case depicts that the point lies inside the body. The lines can be drawn from a grid point along the positive and negative $x$, $y$ and $z$ coordinate directions i.e. six lines can be drawn to mark a grid point. One line is enough to mark a point but other lines can be drawn for confirmation.

Once the grid points are marked, the governing equations need to be modified on the marked grid points to generate the presence of the solid body. Several approaches have been proposed for implementing the accurate boundary conditions for the IBM algorithm. The Discrete Time Derivation (DTD) method of the Brinkman Penalization Method\cite{72} is used in the current work. In this method, the mass and energy equations are left unchanged while the momentum equations are modified on the grid points that lie inside the body. Equation (2.60) represents the governing equations inside and outside the solid body.

\[
\frac{\partial (\rho u_i)}{\partial t} = -\text{Res}_i; \quad \text{Res}_i = \begin{cases} 
\text{Res}_i : \text{non-immersed} \\
0 : \text{immersed}
\end{cases} \quad (2.60)
\]
2.6 Implicit Residual Smoothing

Implicit Residual Smoothing (IRS) is a convergence acceleration method used to speed up the convergence rate of steady-state flow computations. Jameson [73] first used it in 1983 for the Euler equations to increase the stability region of a second-order central difference scheme. Other references include Radespiel et al. [74], Martinelli et al. [75] and Swanson et al. [76].

The discretized Navier-Stokes equation in one-dimension can be written in a general form as:

\[
\left( \frac{\partial u}{\partial t} \right)_i = \text{RES}_i(u) \tag{2.61}
\]

where the spatial derivatives are included in the \( \text{RES}_i(u) \) on the right hand side of equation 2.61. This residual can be approximated with a spatial derivative of \( u \) for example with a second order scheme as:

\[
\tilde{\text{RES}}_i(u) = -a \Delta x \frac{\partial^2 u_i}{\partial t^2} = -a \Delta x \left( u_{i+1} - 2u_i + u_{i-1} \right) \tag{2.62}
\]

The residual at the \( n^{th} \) iteration can be corrected with a predicted change of the pseudo residual:

\[
\text{RES}^n = \text{RES}_n + \left( \tilde{\text{RES}} - \tilde{\text{RES}}^n \right) \tag{2.63}
\]

where \( * \) represents the corrected values. The subscript \( i \) is dropped as the same formulations can be applied at every grid point except at the boundaries. The pseudo-residual is estimated in an implicit way:

\[
\tilde{\text{RES}} = (1 - \alpha)\tilde{\text{RES}}^n + \alpha \tilde{\text{RES}}^{n+1} \tag{2.64}
\]

where, the weighting factor \( 0 \leq \alpha \leq 1 \), and \( n \) and \( n + 1 \) are the current and next time step respectively. The corrected residual then becomes:

\[
\text{RES}^n = \text{RES}^n + \alpha \left( \tilde{\text{RES}}^{n+1} - \tilde{\text{RES}}^n \right) \tag{2.65}
\]
The pseudo-residual at the next time step can be approximated using Taylor’s expansion as:

\[
\overline{\text{RES}}^{n+1} = \overline{\text{RES}}^n + \left[ \frac{\partial \overline{\text{RES}}}{\partial u} \right] \Delta u
\]

(2.66)

Equations 2.65 and 2.66 can be substituted in equation 2.61 to obtain the discretized equation in terms of corrected residual values:

\[
\left[ \left( \frac{1}{\Delta t} - \alpha \frac{\partial \overline{\text{RES}}}{\partial u} \right) \Delta u \right]^n = \text{RES}^n
\]

(2.67)

The pseudo-residual term can be calculated from equation 2.62:

\[
\frac{\partial \overline{\text{RES}}}{\partial u} = -\frac{a}{\Delta x} \partial^2
\]

(2.68)

The subscript \( n \) is removed without any confusion. The residual after the smoothing procedure is equal to \( \overline{\text{RES}} = \Delta u/\Delta t \), which reduces equation 2.67 to:

\[
(1 - \beta \partial^2)\overline{\text{RES}} = \text{RES}
\]

(2.69)

where \( \beta = a a \Delta t / \Delta x \). Since the CFL condition number is defined as \( \sigma = a \Delta t / \Delta x \), \( \beta \) is actually an equivalent CFL number. The smoothing procedure doesn’t affect the final solution as all the residuals go to zero for a converged solution.

A stability analysis of equation 2.69 can be performed to evaluate the efficiency of the smoothing procedure. Since any method can be used to calculate the pseudo-residual, this will result in a new stability region for each method. The second order derivative is used for the smoothing scheme in the current work and the DRP scheme is used to calculate the residual. Therefore, the stability region needs to be calculated for this scheme. The stability constraint is derived using the one-dimensional wave equation and a similar analysis holds in all the three directions. The ratio of the amplitude of the Fourier transformed variable at consecutive time steps, known as the scheme’s amplification factor, is calculated from the numerical
analysis:

\[ G(\omega) \equiv \frac{\hat{u}^{n+1}}{\hat{u}^n} = \sigma \sum_{j=-3}^{3} c_j e^{ij\Delta x} \]  \hspace{1cm} (2.70)

where \( i \) is the square root of negative unity, \( \hat{u} \) is the amplitude of the Fourier transformed variable and \( c_j \) are the coefficients of the DRP scheme. \( j \) goes from -3 to 3 as a seven point symmetric stencil is used in this analysis. Similarly, the amplitude of the residual in wave space after implementing IRS can be calculated by taking the Fourier transform of equation 2.69. The resultant expression is given in equation 2.71.

\[ \text{RES}_{\text{IRS}}(\omega) = \frac{1}{1 + 4\beta \sin^2(\omega h/2)} \]  \hspace{1cm} (2.71)

After using the IRS scheme, the combined amplification factor can be calculated as:

\[ G_{\text{IRS}}(\omega) \equiv G(\omega) \text{RES}_{\text{IRS}} = \frac{\sigma \sum_{j=-3}^{3} c_j e^{ij\Delta x}}{1 + 2\beta - 2\beta \cos(\omega \Delta x)} \]  \hspace{1cm} (2.72)

The scheme stability is based on the value of the CFL number \( \sigma \). To retain the same stability region, the following constraint is imposed:

\[ |G_{\text{IRS}}(\omega)| \leq \sigma \]  \hspace{1cm} (2.73)

Equation 2.73 gives a relation between the smoothing parameter \( \beta \) and the ratio of original and modified CFL numbers \( \alpha^*/\alpha \). The analytical solution is approximated by the simpler form:

\[ \beta > \frac{1}{3} \left( \frac{1}{1.158} \left( \frac{\alpha^*}{\alpha} + 0.1 \right) \right)^2 \]  \hspace{1cm} (2.74)

The equation 2.69 can be extended to three dimensions as:

\[ (1 - \beta_i \partial_i^2)(1 - \beta_j \partial_j^2)(1 - \beta_k \partial_k^2) \text{RES} = \text{RES} \]  \hspace{1cm} (2.75)

The smoothing is implemented successively in three directions and the residual is
replaced with the modified residual after each calculation. To account for large aspect ratio cells, the ratio of the CFL numbers in equation 2.74 is modified using the formulation used in Radespiel et al.[74] for smoothing in ξ direction:

\[
\frac{\sigma^*}{\sigma} \leftarrow \frac{\sigma^*}{\sigma} \frac{1 + \max(r^{1/2}_{\eta\xi}, r^{1/2}_{\zeta\xi})}{1 + \max(r_{\eta\xi}, r_{\zeta\xi})}
\]  

(2.76)

where, \( r_{\eta\xi} = \lambda_{\eta}/\lambda_{\xi} \) and \( r_{\eta\xi} = \lambda_{\eta}/\lambda_{\xi} \). \( \lambda_{\xi} \) is the spectral radius of the Jacobian matrix of the flux terms. Similar forms can be used for the \( \eta \) and \( \zeta \) directions. In the current computation, the residual smoothing is not applied on the boundaries due to stability and computational efficiency constraints.

### 2.7 Multi-Block Topology and Characteristic Interface Boundary Conditions

Since the number of points in the grids used in jet flow calculations is generally several million, it becomes necessary to parallelize the code for faster calculations. The present code is parallelized using MPI 2.0[77] message passing and the CO-COA4 cluster at Penn State is used to perform the computations. More details about the computing resources can be found in Appendix D.

The code is written in Fortran 90 which has a standard MPI library developed for it. The grid needs to be resolved efficiently in different parts of the computational domain to resolve the boundary layer on the nozzle walls and the shear layer developing outside the nozzle exit. Since a structured grid is used in the present research, several grid blocks are used to ease the procedure to obtain the required mesh. A typical multi-block topology around a converging dual-stream nozzle is shown in figure 2.7. The slices are extracted at two axial locations and the grids inside and outside the nozzle are shown in two left and right figure insets. The geometry of the dual-stream nozzle is also shown with core nozzle in blue and fan nozzle in red. The mesh consists of cylindrical and H-type Cartesian blocks.

There are two levels of communication between the processors in the multi-block
Figure 2.3. Multi-block grid generated for dual-stream nozzle: the slices are extracted at two x-locations and the grids inside and outside the nozzle are shown in two left and right figure insets. The geometry of the dual-stream nozzle is also shown with core nozzle shown in blue and fan nozzle shown in red.

grid topology: Inter-block communication, the message passing between local master processors at block interfaces and intra-block communication, the message passing between different processors within a block. One of the processors in each block is designated as the local master processor which gathers the data at the block interfaces from other local child processors to communicate it with local masters of other neighbor blocks after each time step. The local master processors of different blocks communicate with each other and also with local child processors while the local child processors communicate only within the block they belong to. This communication layout are shown in figure 2.4.

The domain decomposition inside the block is performed on the basis of load balancing. The local masters have additional work of communicating the data with other local masters. The parallel performance of the code is discussed in Chapter 3. The intra-block communication is straight forward within the block as there is no difference between the direction and orientation of the grid interfaces. But inter-

block communication between the blocks needs to be carefully implemented to take care of the issues like different grid orientations at the block interfaces and the intersection point of more than two blocks. Another problem with the mesh generation inside the nozzle is the centerline grid singularity when only cylindrical grid is used as shown in figure 2.5(a). The grid point at the centerline has no unique direction of the normal as required for grid transformation.

![Figure 2.4](image)

Figure 2.4. Levels of communication between the processors

(a) Centerline singularity in a polar mesh and (b) Grid singularity at the intersection of polar and Cartesian blocks

The centerline singularity can be avoided by using a H-type Cartesian grid near the centerline, surrounded by four cylindrical grid blocks as shown in figure 2.5(b).
This kind of topology introduces another kind of singularity known as grid singularity where different blocks meet at a point along different directions (see the intersection point of two cylindrical and central H-block). The grid singularity exists where an abrupt change in the slope of a grid line appears. The grid metrics are discontinuous at the singular point because of the discrepancy between the left- and the right-hand limits of the gradients. This means that the grid metrics are not unique at such points. The discontinuity of the grid metrics is an issue to be considered both mathematically and numerically. Figure 2.5(b) inset shows a grid singularity at the intersection of two cylindrical blocks and central Cartesian block.

In conventional CFD, to simplify the calculation, the grid metrics at the singular points are often approximated to single values by averaging the left- and the right-hand limits. This certainly incurs numerical errors from a geometric point of view. In addition, the grid singularity increases the complexity, especially when it comes to discretization, that is, numerical differentiation. It acts like a standing shock in the transformed flux variables, and spurious oscillations can be generated as the numerical differentiation is carried out near the singular points.

In the present work, the characteristic interface condition introduced by Kim and Lee[63] are used to avoid the grid singularity. The computational domain is decomposed into blocks along the singular grid lines, where the left and the right blocks have the left- and right-hand limits of the grid metrics, respectively. The next step is imposing interface conditions at the block interfaces for correct physical communication between the isolated blocks. The interface conditions are derived from the characteristic relations of the compressible Euler or Navier-Stokes equations. The characteristic waves calculated in one block are interchanged or replaced by those in the next block at the interface according to the direction of communication. These conditions ensure correct direction of movement of the flow information at the block interfaces. The formulation of these equations is described in detail in Appendix B.3.
2.8 Non-Matching Mesh Block Interfaces

This feature gives an additional capability to create grids with different grid resolutions in adjacent blocks. This can be used to create finer meshes near the nozzle exit without increasing the computational cost in other regions. This capability is used in generating mesh for the dual-stream nozzle with pylon. The non-matching interface requires data interpolation at the block boundaries before communicating the data between the blocks using the characteristic interface conditions used for matched block interfaces. The Lagrange interpolation method is used to interpolate the flow variables and an iterative procedure is used to locate the coordinates of the grid point on the adjacent block boundary. Details of the interpolation methods applied at the non-matching block boundaries are given in Appendix B.3.2.

2.9 Boundary Conditions

Boundary conditions are very important in a fluid flow simulation as they can alter the solution inside the computational domain by a great extent. In aeroacoustic applications, it should be ensured that the boundaries don’t reflect back the acoustic waves that are usually very weak in nature compared to entropy and vorticity waves. Several boundary conditions are implemented in the EAGLEJet solver tailored for aeroacoustic calculations. The primary boundary conditions include boundary conditions based on method of characteristics, viscous and inviscid walls, Riemann boundary, periodic boundary, total inlet boundary, symmetry boundary and the radiation boundary conditions (See Tam and Webb[62] and Dong[78]).

The method of characteristics boundary conditions[64] have similar derivation as characteristic interface boundary conditions and their details are listed in Appendix B.3. The radiation boundary acts as an non-reflecting boundary by imposing the amplitude of the incoming acoustic wave to be equal to zero. The only waves exiting the domain are acoustic. The entropy and vorticity functions are eliminated to obtain the equations for radiation boundary. These boundary
conditions were initially developed by Tam and Webb\[62\] for perturbations in a uniform mean flow. Information about the mean flow is required to apply these conditions. The equations developed by Tam and Dong\[79\] relax the criterion of uniform mean flow, so these boundary conditions can be implemented without knowing the mean flow field. The details about their implementation in three dimensions can be found in Appendix B.2. The time derivatives are calculated using the equations given in Appendix B.2 at the boundaries and the flow variables can then be calculated.

A buffer zone has been added to the computational domain near the vicinity of the outflow boundaries. The buffer region prevents reflection of the fluctuations which have built up inside the upstream part of the computational domain as a consequence of physical instabilities in the flow. Two types of buffer zones are implemented in the present code to damp the instabilities near the outflow boundaries. In the first method\[53\], the residual terms in the buffer zone are multiplied by a damping function and then the updated conservative variables are calculated. The second method was proposed by Wasistho et al.\[80\] in which the disturbances of all the solution components are gradually reduced to zero within the buffer domain by directly multiplying the disturbances with an appropriate damping function. The first buffer function is calculated as:

\[
\sigma = \frac{1}{2}[1 + \cos(\pi(i - i_{\text{max}} + n_{\text{bp}})/n_{\text{bp}})]
\]  

(2.77)

where \(n_{\text{bp}}\) is the number of grid points in the buffer zone, \(i\) goes from 1 to \(n_{\text{bp}}\), \(i_{\text{max}}\) is the value of grid index on the boundary face. The formulation of the buffer function given by Wasistho\[80\] is given as:

\[
\sigma = (1 - C_1 x_b^2) \left(1 - \frac{1 - \exp[C_2 x_b^2]}{1 - \exp[C_2]}\right)
\]  

(2.78)

with \(0 \leq C_1 \leq 1\), \(C_2 > 0\), buffer domain coordinate \(x_b\) is given as:

\[
x_b = \frac{x_1 - x_s}{x_e - x_s}
\]  

(2.79)
wasistho buffer function is plotted for $C_2 = 10$ and $C_1 = 1.0$ in equation 2.78

$x_s$ and $x_e$ are the values of $x_1$ at the beginning and the end of the buffer domain, respectively. The Wasistho buffer function is plotted in figure 2.6 for $C_2 = 10$ and $C_1 = 1.0$. These buffer functions were tested for the jet cases and the results were compared with the radiation boundary conditions. It was observed that the buffer zone did not provide superior results as compared to the radiation boundary and thus the radiation boundaries were applied at all outer boundaries.

### 2.10 Ffowcs Williams and Hawkings Equation

The Ffowcs Williams and Hawkings (FW-H) equation is an exact rearrangement of the continuity and momentum conservation equations into the form of an inhomogeneous wave equation with two surface source terms and a volume source term as shown in equation 2.80. The FW-H equation is the most general form of the Lighthill’s acoustic analogy because it extends the analogy to include general surfaces in arbitrary motion. The terms in the FW-H equations have a physical meaning and can be better correlated to the actual noise generation mechanisms. The first term on the right hand side of equation 2.80 is the monopole term, the second term is the dipole term and the last term is the quadrupole term which includes the viscous terms. The thickness noise (monopole source) is determined
(a) FW-H integration surface showing two integration patches

(b) Location of the observer defined with respect to jet exit

Figure 2.7. (a) FW-H integration surface showing two integration patches (blue and black dots) shared by one common row of points (red dots). (b) Location of the observer defined with respect to jet exit.

completely by the geometry and kinematics of the body. The loading noise (dipole source) is generated by the force that acts on the fluid as a result of the presence of the body. The classification of thickness and loading noise is related to the thickness and loading problems of linearized aerodynamics. Thus, this terminology is consistent with that of aerodynamics. The quadrupole source term (volume sources) accounts for nonlinear effects, e.g., nonlinear wave propagation and steepening; variations in the local sound speed; and noise generated by shocks, vorticity, and turbulence in the flow field. The FW-H noise prediction code developed by Du[70] is modified for the present work which uses Formulation 1A[49] of FW-H equations 2.84 and 2.85 to make the far-field noise calculations.

In the present implementation, the equations are formulated in terms of pressure rather than the density, and the quadrupole term is omitted. The FW-H variant used by Spalart and Shur[51] is used where density on the acoustic surface is replaced by more generic quantity $\rho^* = \rho_0 (1 + p'/P_0)^{1/\gamma}$. This allows the tapered acoustic data surface to lie closer to the jet plume. The co-flow effect results in a source moving with uniform velocity. This effect can be included while calcu-
lating the noise for jets in flight. Numerical implementations of this formulation have proved to be very robust and efficient and most predictions now use this formulation.

$$\square^2 p'(\vec{x}, t) = \frac{1}{c^2} \frac{\partial^2 p'}{\partial t^2} - \nabla^2 p' = \frac{\partial}{\partial t} [\rho_o U_n] \delta (f) - \frac{\partial}{\partial x_i} [L_i \delta (f)] + \frac{\partial^2}{\partial x_i \partial x_j} [T_{ij} H (f)]$$ (2.80)

where $\square^2$ is the wave operator in three-dimensional space and $f(\vec{x}, t) = 0$ is the moving surface. The equation of the surface ($f$) is defined such that $f > 0$ is outside the body and $f < 0$ is inside it. Equation 2.81 represents the formulation of the monopole source terms and equation 2.82 represents the formulation for dipole or surface forces.

$$U_i = \left( 1 - \frac{\rho}{\rho_o} \right) v_i + \left( \frac{\rho}{\rho_o} \right) u_i \quad (2.81)$$

$$L_i = P_{ij} \hat{n}_j + \rho u_i (u_n - v_n) \quad (2.82)$$

$u_i$ are the components of the local flow velocity vector and $v_i$ are the components of the local surface velocity vector. $u_n$ is the local normal fluid velocity, $v_n$ is the local normal velocity of the body surface and $\hat{n}$ is the unit outward normal vector to surface. $P_{ij}$ is the compressive stress tensor with constant $p_0 \delta_{ij}$ subtracted. Pressure perturbation can be calculated as $p' = c^2 (\rho - \rho_o)$ where $\rho_o$ and $c$ are the density and speed of sound in the undisturbed medium, $\delta (f)$ is the Dirac delta function, $\delta_{ij}$ is Kronecker delta and $H (f)$ is the Heaviside function. $T_{ij} = \rho u_i u_j + P_{ij} - c^2 \rho' \delta_{ij}$ is the Lighthill stress tensor. Expressed in a retarded-time formulation, the acoustic pressure can be divided into two components:

$$p' (\vec{x}, t) = p'_T (\vec{x}, t) + p'_L (\vec{x}, t) \quad (2.83)$$

where the thickness (monopole) component $p'_T$ and the loading (dipole) compo-
nents $p_L' \text{ are given by equations } 2.84 \text{ and } 2.85$.

$$4\pi p'_L(x, t) = \int_{f=0} \left[ \frac{\rho_0 (\dot{U}_n + U_n)}{r(1 - M_r)^2} \right]_{\text{ret}} dS$$

$$+ \int_{f=0} \left[ \frac{\rho_0 U_n \left( r\dot{M}_r + c(M_r - M^2) \right)}{r^2(1 - M_r)^3} \right]_{\text{ret}} dS$$

$$4\pi p'_L(x, t) = \frac{1}{c} \int_{f=0} \left[ \frac{\dot{L}_r}{r(1 - M_r)^2} \right]_{\text{ret}} dS$$

$$+ \int_{f=0} \left[ \frac{L_r - L_m}{r^2(1 - M_r)^2} \right]_{\text{ret}} dS$$

$$+ \frac{1}{c} \int_{f=0} \left[ \frac{L_r(r\dot{M}_r + c(M_r - M^2))}{r^2(1 - M_r)^3} \right]_{\text{ret}} dS$$

These Formulation 1A equations are given in Brentner and Farassat [81]. In these equations, a dot over a variable implies the source time derivative of that variable and the subscript $n$, $r$ and $m$ refer to the dot product with the unit normal vector, the unit radiation vector, or the surface velocity vector normalized by the speed of sound, respectively. In this formulation, integrands with $1/r$ dependence are far-field terms and those with $1/r^2$ dependence are near-field terms. The retarded (source) time $\tau$ can be calculated using the equation $g = \tau - t + |\vec{x} - \vec{y}| / c = 0$ where $t$ is the observer time, $\vec{x}$ is the observer location, and $\vec{y}$ is the source position.

The integration surface is divided into patches of $5 \times 5$ and a two-dimensional Gauss-Lagrange integration scheme is used. Two integration patches are shown in figure 2.7(a) where five points are shared between two patches. One dimensional Lagrange polynomial interpolation method is used to calculate the variables at retarded time. The flow field is extracted from the grid points using Lagrange polynomial interpolation. Three types of data surfaces can be extracted from the code: spherical, cylindrical and a surface coinciding with the grid points. There is no restriction on the number of integration surfaces. Interpolation of the flow
variables is not required if a grid surface is extracted. The contribution from all the surfaces is calculated at the observer location and the pressure time history is obtained. The pressure time history can then be used to calculate the noise spectra in terms of power spectral density at the observer location. The definition of the location of the observer with respect to the jet exit is defined as shown in figure [2.7(b)]. It is assumed that the jet axis coincides with the positive $x$-direction with the jet exit at $x = 0$.

### 2.11 EAGLEJet and PSFWH Solvers Development

The computational procedure to simulate the flow field using EAGLEJet is shown as a flow chart in figure [2.8]. The flow-chart summarizes the key stages of the solver and also lists the layout for writing the solution files for post-processing and noise calculations. The current code is based on the NLDE code originally developed by Erwin [53]. The main features of the code and the details of setting up a case are given as an “User Manual” in Appendix [D].

The FW-H solver known as PSFWH originally developed by Du [70] has been used in the current work for noise calculations. The code has been modified to work with dimensional parameters for the current implementation.

### 2.12 Summary

The present chapter describes the several numerical methods implemented in the EAGLEJet solver for accurate flow and noise simulations. Details of the governing equations in generalized coordinates can be found in Appendix [A]. Several boundary conditions are described in detail in Appendix [B]. The details of using the EAGLEJet solver can be found in Appendix [D]. It can be seen that the numerical methods used in the present work are tailored for accurate noise calculations in aeroacoustic problems. The next chapter discusses some of the validation cases.
used to validate and test the implementation of these numerical methods before performing the more complex jet flow simulations.
Figure 2.8. Process flow-chart for the EAGLEJet RANS-LES solver and PSFWH noise calculation solver.
Navier-Stokes Solver Validation

Jet simulations involve several complexities such as complicated block interfaces, different boundary conditions and governing equations in generalized coordinates. Therefore the different capabilities of the code need to be tested by several benchmark problems before performing the actual jet calculations. Several benchmark cases have been used to validate the different functionalities of the code. Selected validation cases and their results are presented in this Chapter.

3.1 Determining Parallel Efficiency of the Solver

Jet flow calculations require large computational resources and computational time. Thus, to obtain the solution in a time efficient manner, the code should have good scalability such that many processors can be used simultaneously to reduce the computation time. The efficiency of the parallel code depends on the value of speed up that can be achieved by using multiple processors. In the present solver, fine grain parallelism has been implemented and the communication between the processors is performed using MPI 2.0 library subroutines.
The speed up value of a parallel solver can be calculated using equation 3.1:

$$\text{Speed up} = \frac{\text{Wall clock time of sequential execution}}{\text{Wall clock time of parallel execution}} \tag{3.1}$$

The linear or ideal speed up value is equal to the number of processors used in parallel execution but in reality, there is a penalty involved in communicating and synchronizing the data between the processors after each iteration. The overhead increases with number of processors therefore an infinite number of processors will not reduce the execution time to zero.

A two block, three-dimensional grid with a Gaussian pulse in the center of one block (See figure 3.4) is used to test the parallel efficiency of the code. The CPU time elapsed in the parallel part of the code is recorded for each computation and the speed up value is calculated by comparing it with the sequential execution time. The sequential execution time is calculated by subtracting the time taken in the serial part of the code from the total CPU time. Therefore, the execution time for the serial part of the code is removed from the speed up calculation. Table 3.1 summarizes the total wall clock time recorded for serial and parallel calculations for equal time steps with varying numbers of processors.

The speed up values listed in table 3.1 show that the solver has good scalability with increasing number of processors and the observed speedup is reasonably close to the ideal speed up value. It can be seen that a large drop in speed up is observed with 64 processors. A part of this drop is attributed to the communication overhead. This drop suggests that the communication overhead has increased with the number of processors which in turn increases the computation time. Figure 3.1(a) shows the ideal and observed speedup obtained for the same test case. For 64 processors, the computation load for one processor reduces a lot as compared to the communication

<table>
<thead>
<tr>
<th>No. of processors</th>
<th>2</th>
<th>8</th>
<th>16</th>
<th>32</th>
<th>64</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wall clock time (s)</td>
<td>5276.83</td>
<td>1487.72</td>
<td>750.51</td>
<td>350.29</td>
<td>220.22</td>
</tr>
<tr>
<td>Speedup (Eq. 3.1)</td>
<td>1</td>
<td>3.55</td>
<td>6.75</td>
<td>15.07</td>
<td>23.96</td>
</tr>
<tr>
<td>Ideal speed up</td>
<td>1</td>
<td>4</td>
<td>8</td>
<td>16</td>
<td>32</td>
</tr>
</tbody>
</table>

Table 3.1. Parallel Solver Speedup
time between the processors therefore a much larger case should be used to study parallel scaling of the solver for larger number of processors. It should be noted that in the current parallel formulation, the local master in each block has the extra load of gathering and communicating the boundary data with neighboring blocks at each time step.

Another important factor that needs to be considered in large computations is the load balancing. All the processors have to wait until the processor with the maximum computation load finishes its task. Therefore, an efficient load distribution between the processors will reduce the processor idle time and increase the overall speedup. The effect of load balancing is studied by running a 3D one block grid with two processors. Two cases are considered. In the first case the load distribution ratio between two processors is 2:1 and in the second case, the load distribution between two processors is 1:1. The inter-block communication load is same in both the cases.

Table 3.2 summarizes the wall clock time obtained in two cases. It can be seen that the efficient load distribution in case one gives considerable speedup by a factor of 2.02 as compared to inefficient load distribution among the processors in case two. Figure 3.1(b) shows the improvement in the execution time obtained by efficiently distributing the work load among the processors.

### 3.2 Reflection of a Transient Acoustic Pulse by a Wall

The reflection of two-dimensional acoustic pulse by a wall validates the implementation of the no-slip wall boundary condition in the solver. The rectangular domain extends from $-75 < x < 75$ and $0 < y < 75$ with $151 \times 76$ grid points and

<table>
<thead>
<tr>
<th>Load Balance: Inefficient</th>
<th>Efficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wall clock time (s)</td>
<td>9033.52</td>
</tr>
<tr>
<td></td>
<td>4468.24</td>
</tr>
</tbody>
</table>

Table 3.2. 3D Gaussian pulse case: Load balancing, total number of processors used is 2.
Figure 3.1. Solver performance: (a) Code speed up with different number of processors used on a 3D two block Gaussian pulse case. (b) Comparison of execution times for inefficient and optimal load balancing on processors. Total number of processors used is two.

Figure 3.2. Initial pressure disturbance with Gaussian spatial distribution centered at \((x, y) = (0, 20)\) used to model the reflection of an acoustic pulse by a wall. The half-width of the pulse is 3.0 and \(y = 0\) is the solid wall.

the initial pressure disturbance with Gaussian spatial distribution is centered at \((x, y) = (0, 20)\) (See Figure 3.2). The initial conditions are matched with Tam and Dong\[^79\] and the results are compared with the numerical and analytical solution given in Tam and Dong\[^79\].

Figure 3.3(a) shows the pressure perturbations obtained at 100, 300 and 500 time steps with the pulse reflection at \(y = 0\) plane. Figure 3.3(b) show the numerical and exact pressure pulse contours computed at 100, 300 and 500 time steps by
Figure 3.3. (a) Numerical pressure contours at different time steps with a Gaussian pulse centered at \((0, 20)\) and with wall reflection from \(y = 0\) plane. (b) Numerical and exact pressure contours at same time steps given in Tam and Dong [79].

Tam and Dong [79]. The pressure contours are plotted at the same levels for the two cases. At 300 time steps, the pulse has reflected off the wall and at 500 time steps the original pulse has partially moved out of the upper boundary. The results obtained with the present numerical simulations match very well with the exact solution.

Figure 3.4 shows the computed and exact pressure wave forms at different times along the line \(x = y\) (as shown in figure 3.2). The distance measured along this line from the origin is denoted by \(s\). It can be seen that the original pressure pulse and the reflected pulse moves out of the domain with increasing time. Numerical and exact solutions match reasonably well at \(t/\Delta t = 1000\).
3.3 Supersonic Flow Past a Wedge

Supersonic nozzles operating at under-expanded or over-expanded conditions may contain shocks or expansion waves. Therefore, the solver should be able to accurately capture these sharp discontinuities in the flow field, which also requires the appropriate numerical dissipation scheme. To test these capabilities of the solver, a wedge flow is simulated in supersonic conditions to study the formation of detached and attached oblique shocks and the effect of different artificial dissipation switches in capturing the shock waves. Mach 1.5 flow is simulated around a wedge inclined at 5° and 10°. The Mach angle is compared with the Mach-θ-β equation for inviscid supersonic flow. Figure 3.5(a) shows the mesh generated for the 5° wedge. Figure 3.5(b) shows the pressure perturbation contours and the shock wave generated at the wedge corner. The calculated shock angle β for 5° wedge is 47.9° which is close to the observed angle of 49.0°. Figures 3.6 show the shock wave locations for the 10° wedge. The reflected wave can be seen in figure 3.6 wedge due to the symmetry condition imposed on the upper-boundary of the domain. The
Figure 3.5. (a) 201 × 201 grid generated for the 5° wedge. (b) Pressure contours in a steady-state simulation of Mach 1.5 flow around a 5° wedge with an oblique shock angle of 50°.

Figure 3.6. Supersonic flow around 10° wedge: Pressure contours in a steady-state simulation of Mach 1.5 flow around a 10° wedge at non-dimensional time of $tU/L = 4.08$ with a shock angle of 58°.

10° wedge for Mach 1.5 gives an oblique shock as observed with a shock angle of 58° which is close to the actual shock angle of 56.7°.

This case is also used to test the effect of selective artificial dissipation[67] and the high order low pass filter[68]. The supersonic 5° wedge case is executed with (a) sixth order low pass filter, (b) artificial dissipation with a pressure-based switch and (c) artificial dissipation with a pressure- and density-based switch. The pressure data is extracted along the center of the domain at a constant $y$ value and the pressure values are compared for three different numerical dissipation meth-
Figure 3.7. Supersonic flow along $5^\circ$ wedge: dissipation effects (a) Low pass filter, (b) selective artificial dissipation with pressure based switch, (c) selective artificial dissipation with pressure and density based switch and (d) comparison of pressure variation across the shock with different dissipation methods.

The pressure contours for different cases are shown in figures 3.7(a), 3.7(b) and 3.7(c). Figure 3.7(d) shows the pressure variation at constant height along the wedge. It can be seen that the pressure variation across the shock is smoothed with the use of both the low pass filter and artificial dissipation but the smearing is greater in the case of low pass filter. Also the filter is not applied at the boundaries. In the present implementation, the master processor collects the data from slave processors at each time step and filters the data and then distributes the data to the slave processors. This increases the computation time. Artificial dissipation is applied with lower order schemes at the boundary points and each processor applies the damping to its own domain. The pressure based switch gives
better selective dissipation in the domain suitable for jet noise applications and is used in the jet flow simulations. It should be noted that the numerical methods result in overshooting of the flow parameters across the shock and a wavy profile is obtained at both ends of the shock before the flow returns back to its steady value. The exact solution will not have these overshoots.

### 3.4 Gaussian Pulse in a Multi-Block Grid

The Gaussian pulse propagation problem is a commonly used test case for validating aeroacoustic solvers and models as it simulates the propagation of a wave similar to the pressure wave emission from aerodynamic noise sources. The analytical solution for the propagation of sound from a Gaussian pulse is available (See Appendix C.2) for both two- and three-dimensional waves. In the present work, this case is used to validate the governing equations, grid transformation matrices, multi-block grid communication for different inter-block interfaces and the non-reflecting radiation boundary conditions (Dong) for non-uniform mean flow. A pulse, initially Gaussian in nature is located in the domain and propagates out with time. The perturbations in the Gaussian pulse are given by equation 3.2:

\[
F = \varepsilon F_o \exp \left[ -\frac{\ln(2)}{\alpha^2} \left( (x-x_0)^2 + (y-y_0)^2 + (z-z_0)^2 \right) \right]
\]  

(3.2)

where \( F \) can be \( p \) or \( \rho \), \((x_o, y_o, z_o)\) is the location of the center of the pulse, \((x, y, z)\) are the coordinates inside the domain, \( \varepsilon \) is the amplitude of the pulse taken as 1% of the ambient value, \( F_o = \rho_o \) in case of density and is equal to \( P_o/\gamma \) in case of pressure. \( \alpha \) is the half width of the pulse. Three multi-block grid topologies are considered here with the initial Gaussian pulse located at different locations. Figure 3.8(a) shows the initial Gaussian pulse at the center of the five block grid which is typical of the grid topology inside the nozzle. Each block has 60 \( \times \) 60 grid points. This grid tests the block interface communication with a grid singularity at the intersection of three blocks. 10 processors with 2 processors in each block were used in this calculation. A standard Runge-Kutta fourth order scheme is used
Figure 3.8. (a) Gaussian pressure pulse located at the center of the five block mesh at $t = 0$ s. (b) Pressure contours at $t = 9.2$ ms

for unsteady time marching. Figure 3.8(b) shows the pressure perturbations at 9.2 ms. It can be seen that the pulse moves smoothly across the block interfaces and the characteristic interface boundary conditions work well.

Figure 3.9(a) shows the four-block grid with $101 \times 101$ points in each block. The Gaussian pulse is located at the center of one of the blocks. This grid topology tests the multi-block interface where four blocks intersect. This situation arises at the intersection of the blocks inside and outside the nozzle. Figure 3.9(b) shows the pressure perturbations at 0.28 s. It can be seen that the pulse moves smoothly across the intersection and also there is no wave reflection at the boundaries.

Figure 3.10(a) shows another four block grid topology where one block shares its boundary with multiple blocks. The initial Gaussian pulse is located near the boundary of the largest block. Figure 3.10(b) shows the pressure perturbations at 0.28 s where it can be seen that the pressure pulse moves smoothly out of the domain and at the block boundaries.

Figure 3.11(a) shows a 2 block three-dimensional grid with the initial Gaussian pulse at the center of one block. Each block has $61 \times 61 \times 61$ grid points. Figure 3.11(b) shows the pressure perturbations at 2.8 ms where it can be seen that the pressure pulse moves smoothly out of the domain and at the block boundaries.
Figure 3.9. (a) Gaussian pressure pulse located at the center of one of the four blocks at $t = 0$ s. (b) Pressure contours at $t = 0.28$ s.

Figure 3.10. (a) Gaussian pressure pulse located at $(x, y) = (-0.2, -0.2)$ at $t = 0$ s. (b) Pressure contours at $t = 0.28$ s.

Figure 3.12(a) compares the numerical and analytical pressure perturbation for a two-dimensional Gaussian pulse extracted from a four block case at the same time. Figure 3.12(b) compares the numerical and analytical pressure perturbation for a three-dimensional Gaussian pulse extracted from the 2 block case at the same time. Excellent agreement is obtained in both two- and three-dimensional cases.
3.5 Turbulent Subsonic Flow Over Flat Plate

The one-equation Spalart-Allmaras model \cite{57} (S-A model) is implemented in the solver to perform the RANS calculations inside the nozzle. The RANS solution is used inside the nozzle while the eddy viscosity is turned off outside the nozzle and the grid spacing is used to filter the smallest unresolved scales. The Spalart-
<table>
<thead>
<tr>
<th>Mach</th>
<th>Pressure (Pa)</th>
<th>Temperature (K)</th>
<th>Angle of attack (deg.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2</td>
<td>101352.93</td>
<td>294.44</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Table 3.3. Flow conditions for turbulent flat plate with length $L = 5.09$ m

Figure 3.13. Developed turbulent boundary layer profile with Mach 0.2 flow along the flat plate of length $L = 5.09$ m

Allmaras model is validated by computing the RANS subsonic flow over a flat plate and comparing the turbulent boundary layer profile with other turbulence models and experimental data (See Ref. [82, 83]). The plate dimensions and the flow conditions are listed in table 3.3. The grid has one block with $146 \times 91$ grid points and the first grid point off the plate wall lies at $y^+ = 1$.

The developed turbulent boundary layer is plotted in figure 3.13. The boundary layer profile is plotted in $u^+$ and $y^+$ wall coordinates at two locations on the flat plate: near the middle of the plate $x/L = 0.45$ and near the plate trailing edge $x/L = 0.92$ where $L$ is the plate length. The results are compared with other turbulence models and experimental data. Figures 3.14(a) and 3.14(b) show boundary layer development at $x/L = 0.45$ and 0.92 respectively. The log-law (equation 3.3) and the viscous sublayer (equation 3.4) profiles are also plotted. It can be seen in figure 3.13 that the turbulent boundary profile matches well with the experimental data and other turbulence models. The log-law and the viscous sublayer profiles...
are also well resolved.

\[ u^+ = \frac{1}{\kappa} \ln y^+ + C \]  

(3.3)

\[ u^+ = y^+ \]  

(3.4)

where \( C = 5.0 \) and Von Kármán constant \( \kappa = 0.41 \).

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure3.14.png}
\caption{Turbulent boundary layer over flat plate at Mach 0.2 plotted at (a) \( x/L = 0.45 \) and (b) \( x/L = 0.92 \).}
\end{figure}

### 3.6 Subsonic Flow Past a Circular Cylinder

The flow over a cylinder is a useful test case to validate the unsteady flow simulation and DES capabilities in the transition region which occurs between the RANS and LES regions. The cylinder flow is a separated flow case which can test the solver’s capability to determine the point of separation and vortex shedding frequency. This case is relatively easy to set up and has been used to test the following numerical methods in the present code:

- Non-matching interface implementation.
- Immersed Boundary Method implementation.
- RANS capability and DES capability and the behavior of the solution in the RANS and LES regions.

A very small value of the eddy viscosity is to be maintained up to the cylinder’s separating shear layers. High values of vorticity in the shear layers can then lead to the production of high values of eddy viscosity. The boundary conditions, time step size etc., should not affect the solution. The Reynolds numbers considered in this study are \( Re = 150 \) and 90,000. The first \( Re \) falls in the laminar region and the flow at higher \( Re \) is fully turbulent at the separation point.

A validation case was first run at \( Re = 150 \) on a two-dimensional cylinder with body-fitted grid to validate the unsteady part of the code. At low Reynolds number, the Von Kármán vortex shedding occurs at a specific frequency as determined by experiments. A low Reynolds number based on cylinder diameter is chosen where the vortex shedding is in the laminar range. The static pressure is varied to achieve the desired Reynolds number equal to 150 in this case. The static temperature is kept at ambient value of 293.0 K. The static pressure is kept at 3646.2 Pa and the cylinder diameter \( D \) is taken as 1 mm. The mesh contains two blocks with 251 \( \times \) 151 points in each block. The distance of the nearest grid point from the cylinder surface is kept at a wall distance of \( y^+ = 1 \). The simulation is run using dual-time stepping with 60 sub-iterations within one physical time step and implicit residual smoothing is used to accelerate the convergence. The vortex shedding frequency is calculated from the time series plot of pressure. It should be noted that the vortex sheds alternatively from the upper and lower surface of the cylinder. Therefore, the oscillations in the lift force on the cylinder occur at the shedding frequency while the oscillations in drag force occur at twice the vortex shedding frequency. The vortex shedding frequency is calculated as shown in equation 3.5:

\[
 f_s = \frac{StU}{D} \quad (3.5)
\]

where \( St \) is the Strouhal number, \( U \) is the free-stream velocity and \( D \) is the cylinder diameter. For \( 40 \leq Re \leq 200 \), the \( St \) number varies between \( St \sim 0.179 \) to 0.182. The instantaneous pressure contours are shown in figure 3.15 and the variation of pressure with time behind the cylinder at \( (x/D, y/D) = (4, 0) \) and
Figure 3.15. Pressure contours showing vortex shedding on the cylinder at $Re = 150$. Time series data are extracted at points $A$ and $B$ for pressure-time curves.

Figure 3.16. Time series plot of pressure at (a) $(x/D, y/D) = (4, 0)$ behind the cylinder showing drag variation and at (b) $(x/D, y/D) = (0, 4)$ perpendicular to the free-stream flow showing lift variation. Above the cylinder at $(x/D, y/D) = (0, 4)$ are shown in figures 3.16(a)-3.16(b). The time period is calculated to be equal to $8 \times 10^{-5}$ s which corresponds to a $St$ number of 0.18. The vortex shedding frequency has excellent agreement with the expected value.

A similar case with $Re = 90,000$ and Mach 0.2 has been executed with the use of the Immersed Boundary Method (IBM) and non-matching interfaces. Figure 3.17(a) shows the grid with non-matching block interfaces. There are in total 9 blocks. The central red block has the finest grid resolution to resolve the cylinder wall with good accuracy and to keep the $y^+ \simeq 1$ near the cylinder surface. The blue
Figure 3.17. (a) Non-matching grid generated for flow past cylinder at $M = 0.2$. The overlay shows the cylinder boundary and the red block with fine resolution used to resolve the cylinder surface. (b) Mach contours showing vortex shedding over the cylinder with cylinder diameter, $D = 0.019$ m, $Re_D = 90,000$

block has a coarser grid as compared to the red block but a finer grid resolution as compared to the black region to resolve the cylinder wake region and vortex shedding. The black blocks are coarsest with increasing grid spacing away from the cylinder. The diameter of the cylinder is $D = 0.019$ m. Figure 3.17(b) shows the instantaneous Mach contours past the cylinder obtained without the use of any turbulence model.
The pressure coefficient $C_p$ can be calculated on the cylinder surface using equation 3.6:

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2}$$

(3.6)

where $p_\infty$, $\rho_\infty$ and $U_\infty$ are the free-stream pressure, density and velocity values respectively. Figure 3.18(a) shows the variation of the pressure coefficient with time at point A for different governing equations. The “laminar” case represents the unsteady Navier-Stokes mode where no turbulence model is used. It can be seen that the RANS (SA model) simulation gives very small oscillations about at value of $C_p = -1.0$. The DES mode switches to LES away from the wall. The unsteady “laminar” mode shows a similar variation of $C_p$ as obtained in the DES mode. It is clear that the SA model simulation suppresses the unsteady oscillations. The averaged $C_p' = p_{rms}'/(1/2\rho_\infty U_\infty^2)$ values are calculated at point A. The $C_p'$ values for three cases are found to be equal to: $(C_p')_{lam} = 1.64$, $(C_p')_{SA} = 0.95$ and $(C_p')_{DES} = 1.71$. The vortex shedding frequency $St$ is calculated for each case and its values are compared in figure 3.18(b). The DES mode gives the highest vortex shedding frequency but all three values are close to the experimental value of 0.2. The value of $St$ obtained in Cox et al. [84] for similar case with the SA model is approximately 0.21 which is also lowest among all the models used. At $Re = 90,000$, the three-dimensional effects come into play as compared to the flow at $Re = 150$. Therefore, it becomes difficult to match the experimental results with an equivalent two-dimensional simulation. Also, the $Re = 90,000$ case is a transition state just before the drag crisis, so the flow is about to switch from laminar to turbulent and become chaotic.

### 3.7 Transonic Flow Over RAE2822 Airfoil

A steady flow is simulated past an RAE2822 airfoil to test the solver’s capability to simulate flow in the transonic region as is the case for jets operating at high subsonic Mach numbers. The RAE 2822 airfoil is a supercritical airfoil commonly used for the validation of turbulence models. The airfoil problem creates a singularity region near the trailing edge. This case also tests the sensitivity of the numerical
Figure 3.18. (a) Variation of $C_p$ with time on the upper surface of the cylinder shown by point A for “laminar”, RANS and DES simulations. (b) Comparison of vortex shedding frequency ($St$) for “laminar”, RANS and DES simulations, $D = 0.019$ m, $Re_D = 90,000$

method to the grid. The value of eddy viscosity increases in the shear layer which lengthens the shear layer and also reduces the effective Reynolds number. The grid needs to be well resolved near the trailing edge to capture the physics of the flow. The standard Spalart-Allmaras turbulence model is used in the simulations. The airfoil has a chord length of $c = 1.0$ ft. The Reynolds number based on the chord length is equal to $Re_c = 6.5$ million. The free-stream Mach number is 0.729 at an angle of attack of $\alpha = 2.31^\circ$. Table 3.4 summarizes the free-stream conditions for the flow past the airfoil.
Table 3.4. Freestream conditions for flow past RAE2822 airfoil

<table>
<thead>
<tr>
<th>Mach number</th>
<th>Static Pressure(psia)</th>
<th>Temperature(R)</th>
<th>Angle-of-Attack(deg.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.729</td>
<td>15.80734</td>
<td>460.0</td>
<td>2.31</td>
</tr>
</tbody>
</table>

Figure 3.19. (a) Mesh used for simulation of flow past RAE2822 airfoil at angle of attack $\alpha = 2.31^\circ$. (b) Pressure contours in a steady state simulation and the streamlines show the free-stream flow direction.

The mesh has two C-type grid blocks with $185 \times 65$ points in each block as shown in different colors in figure 3.19(a). The physical domain extends up to 27 chord lengths in the axial direction behind the airfoil and up to 25 chord lengths in radial direction on each side. The grid is clustered near the nozzle wall and in the wake region to accurately resolve the flow near the airfoil. Figure 3.19(b) shows the pressure contours near the airfoil surface and it also shows the shock formation on the upper surface of the airfoil. The streamlines show the incoming flow angle with respect to the airfoil chord line. The experimental data is obtained from the RAE 2822 Transonic Airfoil Study # 4 of NASA’s NPARC Alliance Verification and Validation Archive[82]. Figure 3.20 compares the predicted pressure coefficients on the upper and lower surfaces of the airfoil with the experimental[85] and Wind US (SST RANS turbulence model) numerical data. The solver captures the shock location well and also the general trend of the variation of the pressure coefficient on both the upper and lower surfaces of the airfoil. There is some discrepancy observed in the values of the $C_p$ with the experimental data. The possible reasons for this difference are insufficient artificial dissipation and the solution might not
be converged to a steady state. The switch used in artificial dissipation is based on pressure gradient which might not provide sufficient damping near the airfoil surface where flow gradients are highest. The variation in $C_p$ also shows the wiggles across the shock as were observed in supersonic flow past a wedge.

### 3.8 Periodic Pressure Pulse Extraction on Acoustic Data Surface

A one-block uniform mesh with $101 \times 101 \times 101$ grid points is used to test the FW-H solver and to test the extraction of different acoustic data surfaces from EAGLEJet. A periodic pressure pulse is located at the center of the domain and pressure contours at an intermediate time $t$ are shown in figure 3.21(a). A typical FW-H surface is shown by dash lines. Three types of FW-H surfaces are extracted from this domain: cylindrical, spherical and grid surface (which corresponds to a cube in this case). The analytical value of the pressure at an observer location can be calculated for the periodic pulse as shown in Appendix C.1. The pressure data is extracted on these surfaces and the solution is compared with the analytical solution as shown in figure 3.21(b). The computed values match well with the analytical solution for all the grid surfaces.
Figure 3.21. Periodic pressure pulse case for FW-H code validation: (a) Pressure contours showing location of the observer and FW-H surface shown by –– lines. (b) Comparison of pressure perturbations at the observer for different acoustic surfaces with the analytical solution.

3.9 Summary

This Chapter has summarized the benchmark cases performed to validate the various numerical methods implemented in the Navier-Stokes solver. The validation cases are designed to test various features of the code such as parallel efficiency, multi-block grid with non-matching block boundaries, immersed boundary method, inviscid and viscous flow analysis, shock capturing in supersonic flow, unsteady flow analysis, transonic flow, various boundary conditions such as wall conditions, Riemann and radiation boundary conditions. Satisfactory results have been found for all these validation cases. The next Chapter presents single stream nozzle calculations. Several jet cases were run to determine the optimal grid size, boundary conditions and location of FW-H surfaces to obtain reasonably good agreement with the experimental data.
The present chapter describes the flow simulation and noise predictions results performed on a Boeing designed convergent nozzle[35]. The jet flow simulation has been performed at different jet operating conditions and the flow and noise results are compared and analyzed. The nozzle case setup, grid refinement study, the effect of the location of the FW-H surface for data sampling, flow simulation results including boundary layer analysis and statistical analysis of turbulence have been performed and the results are presented in this chapter. The effect of heat on the jet flow is studied and the noise levels are compared for the two cases. The flight effects of the single stream jets are studied by using different co-flow velocities. The numerical results are compared with the available experimental data.

4.1 Introduction

A baseline Boeing-designed convergent nozzle is used in the present work for single stream jet calculations. The nozzle has an exit diameter of $D_j = 2.45 \text{ in.}$ and the nozzle geometry is shown in figure 4.1(a). Unheated and heated calculations are made at jet operating conditions similar to aircraft takeoff conditions. The jet exit Mach number is fixed at $M_j = 0.9$ and two total temperature ratios $TTR = 1.0$
Table 4.1. Convergent nozzle jet operating conditions, $\lambda = M_{cf}/M_j$

<table>
<thead>
<tr>
<th>Case</th>
<th>$M_j$</th>
<th>$TTR$</th>
<th>$\lambda$</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>0.9</td>
<td>1.0</td>
<td>0.0</td>
</tr>
<tr>
<td>II</td>
<td>0.9</td>
<td>1.0</td>
<td>0.13</td>
</tr>
<tr>
<td>III</td>
<td>0.9</td>
<td>1.0</td>
<td>0.22</td>
</tr>
<tr>
<td>IV</td>
<td>0.9</td>
<td>1.0</td>
<td>0.311</td>
</tr>
<tr>
<td>V</td>
<td>0.9</td>
<td>2.7</td>
<td>0.0</td>
</tr>
<tr>
<td>VI</td>
<td>0.9</td>
<td>2.7</td>
<td>0.22</td>
</tr>
<tr>
<td>VII</td>
<td>0.9</td>
<td>2.7</td>
<td>0.311</td>
</tr>
</tbody>
</table>

and 2.7 are used for the unheated and heated jet cases respectively. The co-flow Mach numbers used are $M_{cf} = 0.0, 0.12, 0.2$ and 0.28. Table 4.1 summarizes the jet operating conditions for unheated and heated jets.

Noise calculations are performed at different observer locations of $r/D_j = 72$ and 100 and different inlet angles. The following sections present the nozzle case setup for the EAGLEJet solver, grid refinement study, FW-H surface location study and the flow and noise simulation results for different jet operating conditions. The flight effects are then studied for these cases followed by the analysis of the results.
4.2 Nozzle Case Setup

The mesh generation procedure, initial conditions and the boundary conditions used for the nozzle cases are discussed in this section. The procedure used for running the unsteady flow simulation is also outlined.

Figure 4.2 shows the grid generation inside and outside the nozzle. The grid outside the nozzle is in the shape of tapered funnel and extends up to $60D_j$ downstream of the nozzle exit and $16D_j$ in the radial direction. Figure 4.2 left inset shows the grid topology inside the nozzle. An H-type grid is used at the nozzle center surrounded by four cylindrical blocks. This removes the centerline singularity associated with a pure cylindrical mesh inside the nozzle. However, the present topology does introduce another grid singularity at the intersection of the Cartesian and cylindrical blocks. This is overcome with characteristic interface boundary conditions. Figure 4.2 right inset shows the computational domain layout outside of the nozzle. The 1/3-octave SPL are plotted against Strouhal number $St$ at different inlet angles. The dimensionless frequency, also known as $St$, is defined using the jet velocity $U_j$ and the fully expanded jet diameter $D_j$ as:

$$St = \frac{fD_j}{U_j} = \frac{f}{f_c}$$  \hspace{1cm} (4.1)

where $f_c$ is the characteristic frequency of the jet and effective jet diameter $D_j$ is related to the nozzle exit diameter as:

$$\frac{D_j}{D} = \left[ \frac{1 + (\gamma - 1)M_{jd}^2/2}{1 + (\gamma - 1)M_{jd}^2/2} \right]^{(\gamma+1)/(4(\gamma-1))} \left( \frac{M_d}{M_j} \right)^{1/2}$$  \hspace{1cm} (4.2)

where $M_d$ is the design Mach number. For subsonic jet cases, $D_j$ is nearly equal to the nozzle exit diameter $D$. The jet acoustic Mach number $M_a$ which is different from the jet Mach number $M_j$ is defined as:

$$M_a = \frac{U_j}{a_\infty}$$  \hspace{1cm} (4.3)
where the jet exit velocity is normalized with the free-stream speed of sound $a_\infty$ instead of the local speed of sound. The Reynolds number $Re$ is defined as:

$$Re = \frac{\rho_\infty U_j L_R}{\mu_\infty} \quad (4.4)$$

where $L_R$ is the reference length usually taken as jet exit diameter. The axial grid stretching in the nozzle grids is maintained at about $(\Delta x)_{x/D=25} \sim 1.10 (\Delta x)_{x/D=0}$ and for the radial distance $(\Delta r)_{r/D=3} \sim (1.05 - 1.10) (\Delta r)_{r/D=0}$ after which the grid can be stretched rapidly. The grid resolution near the nozzle exit depends on the highest $St$ number that needs to be accurately resolved. The grid spacing should be sufficiently small to resolve the shortest wave component i.e., highest frequency. The first grid point off the nozzle wall is at approximately $y^+ \sim 20$, which is reasonable since all the details of the boundary layer inside the nozzle are not required to capture the physics of the flow in the shear layer. The relation between the maximum resolvable Strouhal number and the grid spacing is given by the following equation:

$$St_{max} = \frac{cD_j}{N\Delta x U_j} \quad (4.5)$$

where $N$ points are required to resolve the shortest wave component i.e., $\lambda = N\Delta x$. $D_j$ is the effective jet diameter and $U_j$ is the jet exit velocity in the jet core region. For $M_j = 0.9$ jet operating at cold jet conditions, $D_j/D \sim 1$, $U_j/c = 0.9$. It is assumed the $N = 10$ points are enough to resolve the shortest wave length. As a result, if the resolution is desired to reach a highest frequency corresponding to $St = 2$, the estimated grid spacing is:

$$\frac{\Delta x}{D} = \frac{D_j}{D} \frac{c}{NU_j St_{max}} = 1 \times \frac{1}{10 \times 0.9 \times 2} = 0.0556 \quad (4.6)$$

Due to the fact that the temporal and spatial scales of the energy-containing turbulent structures in the mixing layer are gradually growing as they propagate downstream and that the majority of noise sources are confined within two jet potential core lengths from the nozzle exit, it is not optimal to refine the grids uniformly in the whole computational domain according to this rule. Instead,
Figure 4.2. Mesh generated for convergent nozzle with $D_j = 2.45$ in: Computational domain shown with black lines. Two slices extracted at constant axial locations inside and outside the nozzle, total number of grid points in the computational domain = 5.0 million. Left inset: grid topology inside the nozzle with a H-type Cartesian grid in the center surrounded by four cylindrical blocks, Right inset: grid topology outside the nozzle further downstream.

Gradients are gradually stretched in the axial and radial directions. In the azimuthal direction, 120 grid points are used which results in an azimuthal grid resolution of 3 degrees. This circumferential resolution has been found to be acceptable in recent work[70]. The physical time step $\Delta t$ used for unsteady time marching is calculated on the basis of the highest resolvable $St_{\text{max}}$. The formulation for calculating the physical time step is given by equation (4.7):

$$\Delta t_P = \frac{N_P}{2 \times St_{\text{max}} \times f_c}$$  

where $N_P$ is the number of the time steps required to accurately resolve the highest $St_{\text{max}}$ can have a value between 5 to 10.

Total pressure $P_t$ and temperature $T_t$ values are specified at the nozzle inlet to achieve the required jet operating conditions at the nozzle exit. The total conditions are calculated on the basis of the desired jet exit Mach number using
Figure 4.3. Computational domain shown for a nozzle case. Boundary conditions shown on different surfaces and a typical grid FW-H surface shown in grey.

one-dimensional isentropic relations given as:

\[
\frac{P}{P_t} = \left( 1 + \frac{\gamma - 1}{2} M_j^2 \right)^{\frac{\gamma}{\gamma - 1}} \\
\frac{T}{T_t} = \left( 1 + \frac{\gamma - 1}{2} M_j^2 \right)^{-1}
\]  

(4.8)

The typical computational domain showing the boundary conditions and the FW-H surface is shown in figure 4.3. An inviscid (slip) wall boundary condition is used for the first few grid points at the nozzle inlet on both inner and outer walls of the nozzle. A viscous (no-slip) wall boundary condition is used for the rest of the grids points on both inner and outer nozzle walls until the nozzle exit. Radiation boundary conditions are used at the downstream and radial outflow boundaries and Riemann boundary conditions are used at the computational upstream boundaries surrounding the nozzle inlet. These boundary conditions are explained in detail in Appendix B. Initially, uniform flow with ambient pressure and temperature is specified in the computational domain outside the nozzle. A small value of axial velocity \( \sim 1\% U_j \) is specified to remove the numerical error in calculating flow direction at the boundaries. In the heated jet cases, \( TTR \) ratio is used to fix the value of the total temperature. In the co-flow cases, the uniform axial velocity, equal to \( U_{CF} = M_{cf} \times a \), is specified outside the nozzle. The domain inside the
nozzle is initialized with approximately 50% of the jet exit pressure, density and velocity values.

For jet noise calculations, the pressure data is sampled over time on the FW-H acoustic surface. The acoustic surfaces used in the present calculations stretch up to $30 D_j$ downstream of the nozzle exit and extends to $1.5 D_j$ from the jet centerline in the radial direction. These values were chosen after studying four different acoustic surfaces with varying axial and radial closure distances. The observer location with respect to the nozzle exit is given by the radial distance $r$ and the azimuthal inlet angle $\theta$ as shown in figure 4.1(b). The noise calculations are performed for $r/D = 72$ and $100$ and $90^\circ \leq \theta \leq 150^\circ$. The results obtained are compared with the experimental data.

### 4.3 Grid Refinement Study

Two grids with 3.5 million grid points and 5.0 million grid points have been generated to study the effect of grid resolution on the noise levels. To lessen the noise spectra fall at higher frequencies, a finer grid needs to be used near the jet exit. The noise spectra obtained using both grids are compared. Figure 4.4 compares the 1/3-octave SPL at different inlet angles for coarse and fine grids with experiments[86] for a $M_j = 0.9$ unheated jet. Equation 4.9 is used to calculate the 1/3-octave SPL where $n$ is the number of bins available for a center frequency ($f_{cen}$) and $\Delta f$ is the bandwidth. It is observed that the noise spectra fall rapidly at higher frequencies because of the limitations of grid spacing. The coarser grid shows lower SPL levels at higher frequencies as compared to the finer grid. Figure 4.5 shows the noise directivity for the unheated jet with $M_j = 0.9$. It can be seen that the noise levels rise at lower inlet angles for the finer grid as compared to the coarse grid and are closer to the experimental values.

\[
SPL(f)_{1/3-Octave} = \left[ \sum_{f_{low}}^{f_{up}} SPL(f)_{bandwidth} + n \ast 10 \log_{10}(n \Delta f) \right] / n \quad (4.9)
\]
Figure 4.4. 1/3-octave SPL at different inlet angles plotted for coarse (3.5 million) and fine grid (5.0 million) and experiments [86] for $M_j = 0.9$ unheated jet.

Figure 4.5. OASPL (dB) at different inlet angles plotted for coarse (3.5 million) and fine grid (5.0 million) and experiments [86] for $M_j = 0.9$ unheated jet.
4.4 FW-H Surface Location Study

The surface integration in the FW-H equation depends on the assumption that all the noise sources should be enclosed inside the acoustic data surface (ADS) for accurate noise calculations. In jet noise calculations, assumptions have been made that major noise sources are located in the jet plume within $10D_j$ downstream of the nozzle exit. Also the noise generating turbulent flow extends up to $1D_j$ in the radial direction. Different radial and axial closing distances have been studied in the recent publications [22, 17, 51] and the effect of the location of FW-H surface on the noise levels as a function of frequency have been analyzed. The effect of closing the ADS up to the nozzle wall at the nozzle exit and at the downstream end of the ADS have also been studied. In the present work, four different FW-H sampling surfaces are used to sample data for a $M_j = 0.5$ unheated CD jet. The radial and axial closing distances for these surfaces are listed in table 4.2 and these surfaces are shown in figure 4.6(a). As the radial location of the ADS closure is moved away from the nozzle, the grid spacing increases and thus the spectral content at higher frequencies fall more rapidly. But it has to be ensured that most of the noise sources are contained within the ADS for accurate noise calculations. Therefore, a compromise needs to be made between spatial resolution and noise sources enclosure in selecting the radial enclosing distance at the nozzle exit.

Figure 4.6(b) compares the SPL obtained by using the four FW-H surfaces for the CD nozzle (shown in inset in figure 4.6(a)) operating at $M_j = 0.5$. The observer location is $r/D = 72$ and inlet angle $\theta = 150^\circ$. It can be seen that as the radial closure distance of the FW-H surface at the nozzle exit increases, the spectra at the highest frequency falls due to the increase in grid spacing. Surface S1 is too close to the jet plume and fails to include a part of the turbulent eddies within the surface. Hence, the whole spectrum shifts down and the OASPL levels are under-predicted. Surface S3 has the shortest downstream closure distance and under-predicts the OASPL by 2-3 dB. Surface S4 also encloses the turbulent eddies both in radial and axial directions but it looses spectral content at higher frequencies because its located at a larger radial distance from the nozzle exit and hence has larger grid spacing. Surface S2 is able to enclose the noise sources reasonably well at both
Table 4.2. FW-H acoustic surface axial and radial closure locations

<table>
<thead>
<tr>
<th>Surface No.</th>
<th>(r/D)</th>
<th>(x/D)</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1</td>
<td>0.5</td>
<td>29</td>
</tr>
<tr>
<td>S2</td>
<td>1.5</td>
<td>29</td>
</tr>
<tr>
<td>S3</td>
<td>1.5</td>
<td>20</td>
</tr>
<tr>
<td>S4</td>
<td>2.5</td>
<td>29</td>
</tr>
</tbody>
</table>

radial and at downstream ends and has the best available grid resolution and thus used in the present calculations.

4.5 Flow Simulation Results

The flow and noise simulation results for the convergent nozzle are presented next. As mentioned before, the jet flow calculations are performed at the operating conditions listed in table 4.1. The surrounding fluid, initially at rest, is drawn radially in by the flow at a low speed as shown by the streamlines (figure 4.7(a)). This demonstrates that the boundary conditions implemented in the simulation allow the incoming of fluid into the computational domain to feed the flow. Figure 4.7(b) displays the longitudinal evolution of the mean flow rate \(Q_m = \int_S U dy dz\), where \(U\) is the mean axial velocity and \(S\) is the transverse section. Its value grows continuously with \(S\), since the entrainment of the surrounding fluid occurs from the inflow of the computational domain onwards. The linear longitudinal evolution of \(Q\) suggests that the self-similarity region of the jet has been reached in the simulation.

Figures 4.8(a)-4.8(d) show vorticity slices extracted at different axial locations (shown by black vertical lines) for different co-flow Mach numbers. It can be seen at \(x = 0.015D_j\) in figure 4.8(a) that the flow is in quasi-steady state and it goes into transition at \(x = 0.5D_j\) from the steady turbulent boundary layer to the unsteady LES shear layer. The central steady potential core region vanishes between 3\(D_j\)-5\(D_j\) downstream. The shear layer becomes turbulent and eventually the turbulent mixing results in the full jet plume further downstream between 5\(D_j\)-10\(D_j\). It is evident that the transition to turbulence is pushed downstream as co-flow Mach
Figure 4.6. A converging diverging (CD) nozzle operating at $M_j = 0.5$. (a) FW-H surfaces shown: S1 to S4 on non-dimensional pressure gradient (normalized with $\rho_o, c_o$ and $D_j$) contours. (b) Comparison of 1/3-octave SPL obtained for FW-H surfaces S1-S4 at $r/D = 72$ and $\theta = 150^\circ$. 
The turbulent intensity is used to characterize the turbulence level in the shear layer and is calculated using the equation \(4.10\).

\[
TI = \frac{1}{U_j} \left( u'u' + v'v' + w'w' \right)^{1/2}
\]

where \(u'\) is the velocity perturbation in the \(i\)-direction and \(<>\) denotes a time averaged value. The \(Q\)-criterion is generally used to visualize the turbulent eddies. It is defined as:

\[
Q = \frac{1}{2} (\tilde{\omega}_{ij} \tilde{\omega}_{ij} - \tilde{s}_{ij} \tilde{s}_{ij})
\]

where \(\tilde{s}_{ij}\) and \(\tilde{\omega}_{ij}\) are the symmetric and anti-symmetric parts of the velocity-gradient tensor respectively. Figure 4.9(a) shows the \(Q\) iso-surfaces (\(Q = \pm 10^5 s^{-2}\)) for the single stream jet without co-flow. The turbulent eddies can be seen developing downstream of the jet exit. Also shown are the pressure perturbations on the enclosing FW-H surface. Figure 4.9(b) shows the variation of the axial turbulence intensity (calculated using equation \(4.10\)) for different co-flow Mach numbers. The co-flow brings down the gradient in the shear layer which results in reduced turbulence in the jet plume. It can be seen that the turbulence intensity...
Figure 4.8. Vorticity magnitude contours plotted for convergent jet ($D_j = 2.45\ in$) operating at $M_j = 0.9$, $TTR = 1.0$, $M_{cf} = 0.0$, 0.12, 0.2 and 0.28. Slices extracted at different axial locations ($0.015 < x/D_j < 10$) downstream of the nozzle exit to show the transition from laminar to turbulent flow in the shear layer.
levels reach their maxima at about 1.5 times the potential core length and the levels decrease with the increase in co-flow Mach number as expected.

Figures 4.10(a)-4.10(d) show the iso-surfaces of $Q$-criterion for an unheated $M_j = 0.9$ jet at increasing co-flow Mach numbers. The iso-surfaces are plotted at $Q = +10^5 \text{s}^{-2}$ for all the cases. These provide an insight into the evolution of turbulent eddies in the jet plume. It can be seen that the jet plume narrows down with increasing co-flow Mach number and also the eddies are stretched in the axial
direction as parallel stream velocity increases. This behavior is consistent with the vorticity slices shown in figures 4.8(a)-4.8(d).

The turbulent kinetic energy (TKE) per unit mass is calculated using equation 4.12:

$$TKE = \frac{1}{2} \langle u_i' u_i' \rangle$$

Equation 4.12

TKE is the measure of turbulence in the flow and is used to study its evolution in the boundary or shear layer. TKE produced by the large eddies is eventually
dissipated by the viscous forces at smallest scales. Therefore, the highest levels of TKE in a flow field represent the eddies with highest energy content. These high energy eddies are in turn responsible for noise generation in jet flows. Figures 4.11(a) and 4.11(b) show the TKE contours for a $M_{cf} = 0.2$ unheated and heated jets. It can be seen that in case of heated jet, the TKE levels rise as the jet exit velocity increases. The high TKE levels are observed in the shear layer as well as at the end of the potential core where the shear layers merge and produce an uniform turbulent flow. The potential core length shortens in case of the heated flow.

Figure 4.12(a) shows the experimental time averaged axial velocity contours for a 1 in convergent jet operating at $M_j = 0.9$ and $M_{cf} = 0.0$. Figure 4.12(b) shows the numerical data for the same operating conditions for $D_j = 2.45$ in nozzle. It should be noted that the experimental data field starts at $1D_j$ downstream of the nozzle exit. Reasonable agreement is obtained between experimental and numerical data. However, as noted below, the potential core length is under-predicted in the simulations, compared to the PIV data.

Figures 4.13(a) to 4.13(d) compare the time averaged axial velocity contours for cases I to IV listed in Table 4.1. It can be seen that an extended jet potential core is
obtained in the co-flow cases due to the reduced turbulence levels in the jet shear layer caused by the reduced velocity difference across the shear layer. A similar effect can be observed for the centerline mean axial velocity profiles shown in figure 4.14(a). The blue squares show the experimental data without co-flow. The solid lines are plotted using the empirical formula given by Morris\cite{23}. Equations 4.13 and 4.15 represent the empirical formulation of the centerline velocity, $V_c$ and potential core length, $x_c$ respectively\cite{23}.

$$\frac{V_c/V_j - \lambda}{1 - \lambda} = 1 - \exp\left\{\frac{-1}{2\xi_v}\right\} \quad (4.13)$$

$$\frac{\xi_u}{r_j} = 0.04(1 - 0.92\lambda)\frac{x}{r_j} - 0.35 \quad (4.14)$$

$$\frac{x_c}{r_j} = \frac{8.77}{1 - 0.92\lambda} \quad (4.15)$$

Table 4.3 compares the numerical potential core length for different co-flow cases with the empirical formulation\cite{23} as given in equation 4.15. The potential core length is taken as the distance in the axial direction up to which the flow velocity is greater than 95% of the jet exit velocity. The numerical data predicts slightly higher values of the potential core length as compared to the experiments. Figure 4.15(a) compares the variation of the jet potential core length with the values
Figure 4.13. Time averaged axial velocity profiles for convergent nozzle $D_j = 2.45 \text{ in}$ (0.0622 m), $M_j = 0.9$ and $TTR = 1.0$, with parallel stream Mach numbers, (a) $M_{cf} = 0.0$, (b) $M_{cf} = 0.12$, (c) $M_{cf} = 0.2$ and (d) $M_{cf} = 0.28$.

Figure 4.14. Mean centerline (a) and radial (b) axial velocity profiles for single stream convergent nozzle $D_j = 2.45 \text{ in}$, $M_j = 0.9$ and $TTR = 1.0$, with parallel stream Mach numbers, $M_{cf} = 0.0, 0.12, 0.2$ and 0.28. Blue squares show the experimental data[87]. Solid lines show the empirical formulation[23] and dashed lines show the numerical data. $r_{0.5}$ is the velocity based jet half radius i.e., the radial location where $< u > / U_{CL} = 0.5$ and $\delta_0$ is the momentum thickness taken as $0.05D_j$. 
(a) Jet potential core length as a function of co-flow Mach number

(b) Pressure gradient contours with typical FW-H acoustic surface

Figure 4.15. (a) Experimental and numerical values of the jet potential core length for different co-flow Mach numbers, $M_j = 0.9$ and $TTR = 1.0$, $M_{cf} = 0.0, 0.12, 0.2$ and 0.28. (b) Pressure gradient contours (normalized with $\rho_o$, $c_o$ and $D_j$) with FW-H acoustic surface used to sample data for noise calculations.

Table 4.3. Comparison of numerical potential core length values with empirical formula for single stream nozzle $D_j = 2.45$ in, and different co-flow Mach numbers, $\lambda = M_{cf}/M_j$

<table>
<thead>
<tr>
<th>$\lambda$</th>
<th>$(x_c/r_j)_{exp}$</th>
<th>$(x_c/r_j)_{num}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>8.77 (14.0 [87])</td>
<td>9.80</td>
</tr>
<tr>
<td>0.13</td>
<td>9.99</td>
<td>11.00</td>
</tr>
<tr>
<td>0.22</td>
<td>11.02</td>
<td>12.60</td>
</tr>
<tr>
<td>0.311</td>
<td>12.28</td>
<td>13.65</td>
</tr>
</tbody>
</table>
reported in Morris [23] for different co-flow Mach numbers. The experimental and numerical variation of the potential core length with the parallel flow Mach number show a similar trend. A higher value of the potential core length has been reported by Bridges [87] for a similar jet without co-flow as shown in brackets in table 4.3. The potential core length in a free shear layer jet is greatly influenced by the flow conditions at the jet exit. The jet contraction ratio, boundary layer development on the nozzle walls and the excitation method used at the jet exit play an important role in the jet flow evolution. Since, the noise is governed by the large eddies in the jet flow and the noise sources are enclosed within FW-H surface, the difference in the values of numerical and experimental potential core lengths will not affect the far-field noise spectra. The potential core lengths have been under-predicted by other LES and DES simulations as well (See Chapter 6 of Brun et al. [88] and Bodony and Lele [15]). Since, the boundary layer development at the nozzle exit governs the development of the mean flow downstream of the jet exit, the boundary layer thickness is compared for the numerical and experimental data [87] for unheated $M_j = 0.9$ jet.

Figure 4.16(a) shows the radial variation of the mean axial velocity at different axial locations near the jet exit. The experimental data is plotted at $x/D_j = 1.0$. It can be seen that the experimental profile is very close to the numerical variation at $x/D_j = 0.5$. The boundary layer development on the inner and outer nozzle walls for different co-flow Mach numbers are plotted in figure 4.16(b). It is observed that the co-flow has less effect on the inner boundary layer but changes the boundary layer profile on the outer wall. The boundary layer thickens as co-flow Mach number increases. The displacement thickness ($\delta^* = \int_0^\infty \left(1 - \frac{\rho(y)u(y)}{\rho_o u_o}\right) dy$) at the nozzle exit on the inner and outer walls is calculated for different co-flow Mach number as shown in figure 4.17(a). $\rho_o$ and $u_o$ are the free-stream density and velocity values respectively. It can be seen that the co-flow has less effect on the displacement thickness value inside the nozzle but $\delta^*$ grows on the outer wall with increasing co-flow Mach number. The boundary layer profile looks different for the $M_{cf} = 0.2$ case as seen in figure 4.16(b) which results in higher value of $\delta^*$ for this case. It needs to be investigated further. The other cases show a consistent trend. Figure 4.17(b) shows the displacement thickness calculations on the inner
Figure 4.16. (a) Radial mean axial velocity profiles at different axial locations for convergent nozzle operating at $M_j = 0.9$, $TTR = 1.0$, $M_{cf} = 0.0$. (b) Boundary layers on the nozzle walls for $M_j = 0.9$, $TTR = 1.0$, $M_{cf} = 0.0$ to 0.28.

Figure 4.17. Displacement thickness ($\delta^*$) value on inner and outer nozzle walls for jet operating at $M_j = 0.9$, $M_{cf} = 0.0$ to 0.28 (a) $TTR = 1.0$ and (b) $TTR = 2.7$.

and outer walls at the nozzle exit for a heated jet operating at varying $M_{cf}$. It is observed that the co-flow has little effect on the displacement thickness on the inner side of nozzle which is similar to the observation made in unheated jets. The displacement thickness grows with increasing co-flow Mach number which is again consistent with the observations made for unheated jets. Therefore, the parallel stream has similar effect on the boundary layer characteristics of the jet.
Since at realistic operating conditions during takeoff, an engine nozzle exhaust is heated to around 1000 °F (800 K), a heated Mach 0.9 jet provides a better model for a jet engine exhaust flow. The heated flow increases the jet exhaust velocity and thus the acoustic Mach number. The acoustic Mach number plays an important role in noise generation and noise propagation to the far-field. Heated jets are found to be noisier as compared to unheated jets. By simulating a heated jet, the effect of flow temperature on the jet noise can be captured and studied. The effect of heating on the jet noise is studied through two jets operating at unheated and heated jet conditions with jet exit Mach number of \( M_j = 0.9 \) and total temperature ratio of \( TTR = 1.0 \) and 2.7. The acoustic Mach number is subsonic for the unheated jet and supersonic for the heated jet with \( M_a = 1.39 \). A rise in noise levels has been observed at all observation angles in the heated jet case. The length of the thermal jet core, \( x_{CT} \) is defined as the distance from the nozzle exit to the point where the relative normalized temperature, \( \Theta_{CL} = (T_{CL}/T_0 - 1)/(T_j/T_0 - 1) \) falls to 0.98. The lengths of thermal and potential jet cores are compared for the unheated and heated jets and are subsequently compared with the experiments. It is observed that the length of the potential jet core decreases with an increase in jet exit temperature and the dynamic jet core length is larger than the thermal jet core in heated jets. The jet heating causes a turning of the shear layers towards the jet axis. Also in line with experiments, the simulations predict a faster damping of turbulence in the hot jets.

The jet Mach number can be defined as:

\[
M = \frac{V_j}{a}
\]

(4.16)

where \( a = \sqrt{\gamma RT} \), \( \gamma \) is the ratio of specific heats and taken to be equal to 1.4 for air and \( V_j \) is the jet exit velocity. \( R \) is the gas constant and is equal to 287.0 \( J/(kgK) \). \( T \) is the static jet exit temperature in absolute units. From equation 4.16, it follows that for a constant Mach number, \( V_j \) will increase in proportion to \( T^{1/2} \). Since jet noise is proportional to the 8th-power of \( V_j \) it seems that the jet noise should increase as \( T^4 \) (again, assuming a constant Mach number). Temperature also impacts the Reynolds number of the jet. The Reynolds number is a non-
dimensional parameter that partially characterizes the amount of turbulence in a jet flow and whether the boundary layer of the flow is turbulent, transitional or laminar. In general, as the Reynolds number of the jet increases the turbulence in the jet increases. As shown before, $V_j$ increases with increasing temperature, which in turn increases the Reynolds number. However, the kinematic viscosity also increases with increasing temperature and more rapidly than $V_j$. Thus, as the temperature is increased the Reynolds number of the jet decreases. As the Reynolds number of the jet decreases, the turbulence in the jet decreases and if the Reynolds number decreases far enough, it is possible for the jet nozzle’s boundary layer to change from turbulent to transitional or laminar. This is known as a “Reynolds number effect”.

A total temperature ratio ($TTR$) of 2.7 is used to study the effect of heat on the core flow. The heated flow increases the local speed of sound and local jet speed and thus the effective acoustic Mach number. It has been demonstrated that the noise characteristics of the jet flow depends on its acoustic Mach number and this effect increases as the acoustic Mach number becomes supersonic.

Figure 4.18(a) shows the mean axial velocity contours for an unheated jet and 4.18(b) shows the similar contours for heated jet. It can be seen that the potential core length shortens in the heated jet. Figure 4.18(c) shows the relative mean temperature contours in the heated jet. The thermal potential core is shorter than the dynamic potential core as evident from figures 4.18(b) and 4.18(c). The ratio of the dynamic to thermal potential core is now studied for different jet operating conditions.

Cases V to VII from table 4.1 are used to evaluate the effect of heating on the jet characteristics. Figure 4.19 compares the numerical centerline mean axial velocity for unheated and heated jets with $M_{cf} = 0.2$ with the Witze’s empirical formulation. The Witze’s formulas used for the centerline velocity are listed in equations 4.17-4.19. The potential core length reduces in the heated jet and the turbulence intensity levels rise. The jet potential core lengthening is observed in the heated jet in a parallel stream which is consistent with the unheated jet results. The maximum turbulence intensity value falls in unheated jets with a
Figure 4.18. Convergent nozzle $M_j = 0.9$, $TTR = 1.0$ and 2.7: (a) jet mean axial velocity contours for cold jet, (b) jet mean axial velocity contours for heated jet, and (c) jet mean temperature contours for heated jet plotted against the axial distance.

parallel stream. But in the case of the heated jets, little change in the maximum centerline turbulence intensity is observed between different co-flow Mach number cases.

\[ u_c(\bar{x}) = 1 - \exp\left(\frac{-1}{\kappa\bar{x}(\bar{\rho}_e)^{0.5} - X_c}\right) \]  

\[ \bar{x}_{core} = 0.70/\kappa(\bar{\rho}_e)^{0.5} \]  

where $\bar{x} = x/r_j$, $X_c = 0.70$ and $\bar{\rho}_e = \rho_e/\rho_j$

\[ \kappa = 0.08(1 - 0.16M_j)(\bar{\rho}_e)^{-0.22} \]

Figure 4.20(a) shows the self-similar behavior of the jet flow in the developing (within the potential core) and developed flow region (beyond the potential core) for a heated jet with $M_j = 0.9$. The radial relative temperature variation is plotted
Figure 4.19. Centerline mean axial velocity (left y-axis) and centerline turbulence intensity levels (right y-axis) for single stream nozzle, $M_j = 0.9$ and $TTR = 1.0$ and 2.7, $M_{cf} = 0.0$ and 0.2.

Table 4.4. Comparison of jet flow characteristics for unheated and heated single stream nozzle $D_j = 2.45$ in, $M_j = 0.9$, $TTR = 1.0$ and 2.7, $T_j = 680.81$ K and $M_{cf} = 0.0, 0.12, 0.2$ and 0.28, Witze [89] potential core length calculated using equation 4.18. The values given in brackets are reported by numerical simulations [22, 17] and experiments [90].

<table>
<thead>
<tr>
<th>$TTR$</th>
<th>$M_a$</th>
<th>$M_{cf}$</th>
<th>$x_{CU}/D_j$</th>
<th>$x_{CT}/D_j$</th>
<th>$x_{CT}/x_{CU}$</th>
<th>$(u'<em>{CL})</em>{max}/U_j$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>0.9</td>
<td>0.0</td>
<td>4.9(4.8)[89]</td>
<td>-</td>
<td>-</td>
<td>0.15(0.17)[23]</td>
</tr>
<tr>
<td>1.0</td>
<td>0.9</td>
<td>0.12</td>
<td>5.5</td>
<td>-</td>
<td>-</td>
<td>0.14 (0.16)</td>
</tr>
<tr>
<td>1.0</td>
<td>0.9</td>
<td>0.2</td>
<td>6.3</td>
<td>-</td>
<td>-</td>
<td>0.13 (0.15)</td>
</tr>
<tr>
<td>1.0</td>
<td>0.9</td>
<td>0.28</td>
<td>6.83</td>
<td>-</td>
<td>-</td>
<td>0.11 (0.13)</td>
</tr>
<tr>
<td>2.7</td>
<td>1.39</td>
<td>0.0</td>
<td>3.8(2.5-3.0)[89]</td>
<td>3.08</td>
<td>0.81(0.86)</td>
<td>0.16</td>
</tr>
<tr>
<td>2.7</td>
<td>1.39</td>
<td>0.2</td>
<td>4.4</td>
<td>3.9</td>
<td>0.86</td>
<td>0.15</td>
</tr>
<tr>
<td>2.7</td>
<td>1.39</td>
<td>0.28</td>
<td>5.05</td>
<td>4.4</td>
<td>0.871</td>
<td>0.15</td>
</tr>
</tbody>
</table>

at different axial locations ($3 < x/D_j < 15$). The radial distance is normalized with respect to the jet half-radius based on the jet exit temperature. Figure 4.20(b) shows the similar behavior in radial mean axial velocity profiles normalized with jet centerline velocity. The radial distance is normalized with the jet half-velocity radius and vorticity thickness. It is observed that two self-preserving regions can be identified: one upstream and one downstream of the potential core. The behavior is preserved even at $x/D_j = 15$. 

Figure 4.20. Radial profiles of the relative temperature (a) and mean axial velocity (b) showing jet similarity inside and outside potential core region, \( M_j = 0.9 \) and \( TTR = 2.7 \). Radial distance normalized with half temperature radius \( r_{1/2,T_{CL}} \), half velocity radius \( r_{1/2,U_{CL}} \) and vorticity thickness \( \delta_\omega \).
Table 4.4 summarizes the jet flow characteristics of the unheated and heated jet. The values given in brackets are reported by other numerical simulations\[22, 17\] and experiments\[90\]. There are some trends that are similar in both unheated and heated jets, but there are few differences as well. The length of the dynamic potential core increases with co-flow Mach number and this trend is also found in heated jets. The thermal potential core length also increases with co-flow Mach number. The ratio of thermal to dynamic potential core increases gradually with the co-flow Mach number in the heated jet. This ratio is found to be similar to the value obtained by Shur et al.\[17\]. The centerline maximum axial turbulence intensity levels (\((u'_{CL})_{max}\)) decrease with co-flow Mach number in unheated jets. But in the case of heated jets, there is little change in \((u'_{CL})_{max}\) values across different co-flow Mach numbers. A probable reason for this observation is the high shear developed between the jet flow and the surrounding flow due to elevated jet exit velocity in the heated jet. This high gradient increases the centerline turbulence intensity and does not vary much in the presence of the co-flow. Especially no difference is observed for the \(M_{cf} = 0.2\) and 0.28 cases where the parallel flow velocity difference is just \(\Delta M_{cf} = 0.08\).

Equations 4.20 and 4.21 are used to calculate the similarity coordinate and the vorticity thickness used in figures 4.20(a)-4.21(f).

\[
\eta = \left( r - r_{0.5} \right)/\delta_\omega \tag{4.20}
\]

\[
\delta_\omega = U_j/\max|\partial \bar{u}/\partial r| \tag{4.21}
\]

where \(\max|\partial \bar{u}/\partial r|\) is the maximum value of the radial velocity gradient and \(r_{0.5}\) is the jet half velocity radius.

Figure 4.21(a) shows the radial variation of the axial turbulence intensity at different axial locations. The jet exit velocity is used to determine the jet half velocity radius in this case. Even though the final two axial locations are downstream of the end of the potential core, the use of the jet centerline velocity, rather than the jet exit velocity provides an improved collapse of the peaks of the turbulence intensity profiles. Similar higher values observed in the experiments at higher downstream locations. The similarity coordinate is the radial distance normalized with the
(a) Radial variation of axial turbulence intensity, $M_j = 0.9$

(b) Radial variation of axial turbulence intensity, $M_j = 0.9$

(c) Radial variation of axial turbulence intensity, $M_j = 0.25$

(d) Radial variation of mean axial velocity, $M_j = 0.9$

(e) Radial variation of mean axial velocity, $M_j = 0.9$

(f) Radial variation of mean axial velocity, $M_j = 0.25$

Figure 4.21. Radial profiles of the mean axial velocity and axial turbulence intensity at different axial locations. Radial similarity coordinate equation (4.20) on x-axis calculated using jet half velocity radius and normalized with vorticity thickness calculated using equation (4.21).
axial mean half velocity radius and local vorticity thickness. Figure 4.21(b) shows similar values but jet centerline velocity is used to determine the jet half velocity radius. It is observed that the peaks of turbulence intensity curves collapse better in the second case. A similar behavior is observed in the experimental data plotted for $M_j = 0.25$ unheated jet in figure 4.21(c). The turbulence intensity levels are higher for the $M_j = 0.9$ jet as compared to the measurements that were performed on a low Mach number jet $\sim 0.25$. At larger axial distances, the $u'$ levels are lower at the similarity coordinate $\eta = -1.0$ in both numerical and experimental data.

Figure 4.21(d) shows the radial variation of the mean axial velocity at different axial locations. Here, the jet exit velocity is used to normalize both mean axial velocity and jet half velocity radius. It can be seen that the collapse of the axial velocity profiles is not as good for values of $\eta < -0.5$ because the jet is not in the potential core region for $x/D_j > 5.0$. On the other hand, figure 4.21(e) uses the jet centerline velocity to normalize both $x$- and $y$- axis quantities and a much better collapse has been obtained. This shows that the jet is self-similar relative to its centerline velocity values. Figure 4.21(f) shows similar radial variation of the axial velocity at different axial locations for the experimental data.

Figure 4.22(a) shows the axial variation of the centerline mean temperature for heated jets with $M_{cf} = 0.0$ and 0.2. The axial distance is normalized with respect to the thermal potential core length. It is observed that the temperature decay has a higher gradient in the case of $M_{cf} = 0.2$.

Figure 4.23(a) compares the mean centerline axial velocity profiles for the heated and unheated jet with the Witze empirical relation given for both cold and heated jets by Witze[89]. The velocity is plotted against the axial distance normalized with the numerical jet potential core. It can be seen that both heated and cold numerical profiles match well with the empirical formulation. Figure 4.23(b) shows the mean centerline axial velocity profiles as a function of the axial distance normalized by the Witze parameter used to calculate the empirical core length. The profiles collapse well with each other. There is good collapse observed using normalization with both numerical and empirical core lengths.

Figure 4.24 compares the jet spread rate and the inverse of centerline mean axial
Figure 4.22. Convergent nozzle $M_j = 0.9$ and $TTR = 1.0$ and 2.7: (a) jet centerline relative temperature profile plotted against the axial distance normalized with the thermal potential core length defined as the axial location where temperature falls to 0.98 times its jet exit value, $M_{cf} = 0.0$ and 0.2. (b) jet centerline axial turbulence intensity values plotted against the axial distance normalized with the dynamic potential core length defined as the axial location where axial velocity falls to 0.98 times its jet exit value.

Figure 4.23. Convergent nozzle $M_j = 0.9$ and $TTR = 1.0$ and 2.7: (a) Jet centerline mean axial velocity values plotted against the axial distance normalized with the dynamic potential core length. Witze empirical formulation is also plotted. (b) Jet centerline mean axial velocity values plotted against the Witze parameter [89] which is used to calculate the potential core length.
velocity for cold and heated jets. It can be seen that the jet half radius is similar for cold and heated jets while the axial velocity decays faster in the heated jet.

4.6 Statistical Analysis of Turbulence for Jets in Flight

Several statistical quantities can be used to quantify the effects of parallel stream on the jet. Three cases I, V, and VI from table 4.1 are used to study the turbulence statistics in the jet shear layer. One of the measures of the turbulence properties of the flow is the two-point correlation function that relates the movement of turbulence downstream of the nozzle to that at a fixed location in the flow field. The two-point cross-correlation function between two fluctuating signals $u_i$ and $u_j$ is defined as:

$$R_{ij}(\vec{x}, \vec{\eta}, \tau) = \overline{u_i(\vec{x}, t)u_j(\vec{x} + \vec{\eta}, t + \tau)}$$  \hspace{1cm} (4.22)$$

where the over-bar denotes the time averaged value, $u_i$ is the velocity fluctuation
in the \(i^{th}\) direction, \(\vec{\eta}\) denotes the separation distance between two probes and \(\tau\) is the time delay of two signals. The correlation function quantifies the dependence of the signals on one another. Physically, this result implies that the turbulent processes producing and dissipating energy at the different scales of motion, as well as transferring energy between the different scales of motion, are in equilibrium as the flow evolves downstream. The relation between the autocorrelation function and the spectral density of the two signals is given by equation 4.23:

\[
R(t) = \int_{-\infty}^{\infty} S(f) \exp(2\pi if t) df
\]

(4.23)

where \(S(f)\) is the spectral density calculated by Equation 4.24:

\[
S(f) = \lim_{T \to \infty} \frac{1}{T} \langle X(f)X^*(f) \rangle
\]

(4.24)

where \(X(f)\) is the Fourier transform of \(x(t)\) and \(X^*(f)\) is the complex conjugate of \(X(f)\). The equations which relate the spectral density and the auto-correlation are called the Wiener-Khintchine equations. The dimension of auto-correlation function is the \((\text{dimension of the signal})^2\). The convection velocity \(U_c\) can then be calculated by joining time delay to the peaks of the correlation coefficients at different separation distances. Data are sampled at different locations in the shear layer and also in the radial direction at several axial locations and the correlation coefficients are calculated as a function of separation distance.

The fourth order correlations give information about the Reynolds stress tensor, which is responsible for the noise generation in viscous jet flows. This information is used while calculating the acoustic field using the Lighthill’s analogy. The fourth order correlation decays more rapidly with distance as compared to the second order correlation. The fourth order two-point cross correlation function can be calculated from equation 4.25 as:

\[
R_{ijkl}(\vec{x}, \vec{\eta}, \tau) = u_i u_j(\vec{x}, t) u_k u_l(\vec{x} + \vec{\eta}, t + \tau)
\]

(4.25)

The auto-correlation coefficient for the axial velocity can be calculated by setting
Another method of calculating the fourth order function is by taking out the mean part from the velocity fluctuations as:

\[ R_{ijkl}(\vec{x}, \vec{\eta}, \tau) = (u_i u_j(\vec{x}, t) - \langle u_i u_j \rangle)(u_k u_l(\vec{x} + \vec{\eta}, t + \tau) - \langle u_k u_l \rangle) \]  

(4.26)

This definition is used in the present work. The fourth order cross-correlation coefficient can now be calculated as:

\[ r_{ijkl}(\vec{x}, \vec{\eta}, \tau) = \frac{R_{ijkl}(\vec{x}, \vec{\eta}, \tau)}{\sqrt{\langle u_i u_j \rangle^2 \langle u_k u_l \rangle^2}} \]  

(4.27)

The quasi-normal hypothesis is generally used to relate the fourth and second order correlations as given in equation 4.28.

\[ R_{ijkl} = R_{ij} R_{kl} + R_{ik} R_{jl} + R_{il} R_{jk} \]  

(4.28)

Lighthill (See Morris and Zaman[91]) showed that, if the mean square turbulent velocity fluctuations and the flatness factor are independent of the separation distance, then the fourth order auto-correlation coefficient can be related to the second order auto-correlation coefficient by,

\[ r_{aaaa}(\vec{x}, \vec{\eta}, \tau) = r_{aa}^2(\vec{x}, \vec{\eta}, \tau) \]  

(4.29)

This formulation is used to compare the correlation values with the direct calculation in figures 4.26(a)-4.26(c). When the quasi-normal hypothesis is not applied, equation 4.29 becomes,

\[ r_{aaaa}(\vec{x}, 0, \tau) = 1 + (T_{\alpha}(x) - 1)r_{aa}^2(\vec{x}, \vec{\eta}, \tau) \]  

(4.30)

where \( T_{\alpha}(x) \) is the flatness factor. At zero separation and time delay and for \( i = j = k = l = \alpha \), Equation 4.27 reduces to[91],

\[ r_{aaaa}(\vec{x}, 0, 0) = \frac{u_{\alpha}^4(\vec{x})}{\left[u_{\alpha}^2(\vec{x})\right]^2} \]  

(4.31)
This is the flatness factor for the $u_\alpha$ velocity fluctuation which is denoted by $T_\alpha(x)$. It is required in equation 4.30 when the quasi-normal hypothesis is not used. Figures 4.25(a)-4.25(c) show the second order two point correlation coefficients in the shear layer $y/D_j = 0.5$ with pivot at $x/D_j = 5.0$. The peak correlation value drops as the separation distance $\xi$ increases. The eddy convection velocity can be calculated from the time delays to the peaks of the correlation curves. It is observed that the convection velocity decreases in the heated jet and with a co-flow. Figures 4.25(d)-4.25(f) show the fourth order correlation coefficient calculated using equation 4.26. The fourth order correlation decays more rapidly with distance as compared to the second order correlation values. These functions are used to calculate the length and time scales of the turbulent eddies in the jet flow.

Figures 4.26(a)-4.26(c) show the second and fourth order spatial correlation coefficients as a function of separation distance between the reference probe and other probes at zero time delay. It can be seen that the coefficient values get lower with increasing separation distance. The curve is also plotted using equation 4.29. The cold and heated jets have similar correlation values variation with separation distance. The fourth order correlation values are higher compared to equation 4.29 for the no co-flow cases. But the fourth order coefficients are lower for the heated jet with co-flow and are very close to the square of the second order correlation values. Figures 4.26(d)-4.26(f) show the variation of time delay for maximum cross correlation with separation distance for cold and heated jets with and without co-flow. Both second order and fourth order correlation values are plotted. The linear fits are drawn along both the curves. It is observed that the slope of the linear fits reduces in the heated jet as compared to a similar unheated jet case. The slope reduced further for the heated jet case with co-flow.

The radial variation of the flatness factor, $T_1$, is shown in figures 4.27(a)-4.27(c). The radial distance is shown relative to jet half radius. It is observed that the flatness factor lies between 3.0 – 4.0 inside the jet lip line but increases to a very high value outside the jet in heated jet case with co-flow.

The frequency dependent phase speed can be calculated from the phase of the
Figure 4.25. Convergent nozzle $M_j = 0.9$ and $TTR = 1.0$ and 2.7: (a) Second order two point correlation coefficient for unheated jet. Red lines show the envelope joining the peak of correlation coefficients. (b) Two point correlation coefficient for heated jet. (c) Two point correlation coefficient for heated jet with $M_{cf} = 0.2$. (d) Fourth order two point correlation coefficient for unheated jet. (e) Fourth order two point correlation coefficient for heated jet. (f) Fourth order two point correlation coefficient for heated jet with $M_{cf} = 0.2$. $\xi$ is the separation distance between two probes. Correlations plotted in the shear layer $y/D_j = 0.5$ at $x/D_j = 5.0$. 
Figure 4.26. Convergent nozzle $M_j = 0.9$ and $TTR = 1.0$ and 2.7: (a) Correlation coefficients at zero time delay for unheated jet. (b) Correlation coefficients at zero time delay for heated jet. (c) Correlation coefficients at zero time delay for heated jet with $M_{cf} = 0.2$. (d) Variation of time delay for maximum cross correlation with separation distance for unheated jet. (e) Variation of time delay for maximum cross correlation with separation distance for heated jet. (f) Variation of time delay for maximum cross correlation with separation distance for heated jet with $M_{cf} = 0.2$. $\xi$ is the separation distance between two probes. Correlations plotted along the lip line $y/D_j = 0.5$ at $x/D_j = 5.0$. 
complex cross spectra as:

\[ U_c(f) = 2\pi f / \left( \frac{d\Phi(f)}{d\xi} \right) \quad (4.32) \]

where \( \Phi(f) \) is the phase delay of the complex cross-spectrum at a downstream location with a separation distance \( \xi \) relative to the fixed probe. Figures 4.27(d)-4.27(f) show the phase velocity plotted against Strouhal number in unheated and heated jets with and without co-flow. The general logarithmic function curve fits given by Morris and Zaman[91] and Harper-Bourne[92] are also shown for comparison. It is observed that the unheated jet deviates from the logarithmic curve most while the heated jet with and without co-flow have more data near the experimentally fitted curve. This scatter in data is primarily due to wiggles in the phase delay at different separation distances. A longer sampling time would smooth the phase delay and provide a more uniform phase velocity.

The time and length scales, denoted as \( L_\tau \) and \( L_x \) respectively, of the turbulent structures quantify the turbulence decay rate and spatial extent in the shear layer. The turbulence axial length scale can be estimated by determining the variation of coherence amplitude with separation distance[91]. The length scale is set to the 1/e decay point (separation distance) in the decay of the amplitude of the cross-correlation functions at zero time delay. The time scale is defined as the 1/e time delay point in the envelope of the space-time correlation functions. The length and time scales obtained from two-point correlation values for cases I, V and VI in the lip line are summarized in the table 4.5. It is observed that the convection speed of the turbulent eddies drops when the jet is heated and further lowers in the heated jet with co-flow. The time scale calculated from the 1/e time delay point of the auto-correlation is denoted by \( L_{\tau,\tau_1} U_j / D_j \) and the time scale calculated from the 1/e time delay point in the envelope of the space-time correlation functions is denoted by \( L_{\tau,\tau_1} U_j / D_j \). The length scale calculated from second order correlation functions is denoted by \( L_{x,\tau_1}/D_j \) and the one calculated from fourth order correlation functions is denoted by \( L_{x,\tau_1\tau_1}/D_j \).

It is observed that the time scales \( L_{\tau,\tau_1} U_j / D_j \) and \( L_{\tau,\tau_1} U_j / D_j \) increase in the heated jet and increase further in the heated jet with co-flow. A higher value
Figure 4.27. Convergent nozzle $M_j = 0.9$ and $TTR = 1.0$ and 2.7: (a) Radial variation of flatness factor, $T_1(x)$ for unheated jet. (b) Radial variation of flatness factor, $T_1(x)$ for heated jet. (c) Radial variation of flatness factor, $T_1(x)$ for heated jet with $M_{cf} = 0.2$. The reference probe is fixed at $x/D = 5.0$. Probe moved along the radial direction. (d) Convection velocity as a function of Strouhal number for unheated jet. (e) Convection velocity as a function of Strouhal number for heated jet. (f) Convection velocity as a function of Strouhal number for heated jet with $M_{cf} = 0.2$. 1: Morris and Zaman [91], 2: Harper-Bourne [92].
Table 4.5. Predicted length and time scales, and convection speed of the turbulent structures of the cold and heated jets operating at $M_j = 0.9$. Calculated at $x/D_j = 5.0$ along the lip line based on second order cross-correlation of the axial velocity fluctuations.

<table>
<thead>
<tr>
<th>$M_a$</th>
<th>$M_{cf}$</th>
<th>TTR</th>
<th>$L_{T_{r11}}U_j/D_j$</th>
<th>$L_{T_{r11}}U_j/D_j$</th>
<th>$L_{x_{r1111}}U_j/D_j$</th>
<th>$U_c/U_j$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.9</td>
<td>0.0</td>
<td>1.0</td>
<td>0.37426</td>
<td>1.1856</td>
<td>0.22933</td>
<td>0.78</td>
</tr>
<tr>
<td>1.39</td>
<td>0.0</td>
<td>2.7</td>
<td>0.64081</td>
<td>1.9866</td>
<td>0.23035</td>
<td>0.42</td>
</tr>
<tr>
<td>1.39</td>
<td>0.2</td>
<td>2.7</td>
<td>1.2566</td>
<td>3.4106</td>
<td>0.16404</td>
<td>0.25</td>
</tr>
</tbody>
</table>

means that the eddy decay rate is slower in flows with a parallel stream which is consistent with the lower levels of peak turbulence in high co-flow cases. The length scales based on second order correlation decrease in the heated jet as compared to the unheated jet but a higher value is obtained for a jet with a parallel stream. The opposite behavior is obtained with the fourth order length scale which reduces in the heated jet with co-flow as compared to cold and heated jets without co-flow. The convective velocity decreases in heated jets as compared to unheated jets and decreases further in the heated jet with co-flow.

### 4.7 Noise Predictions and Forward Flight Analysis

Noise predictions are made for the unheated and heated jets using the FW-H equation described in Chapter 2, Section 2.10. Two observer locations at $r/D_j = 72$ and 100 are chosen that are in true far-field for both cold and heated subsonic jets. The polar or inlet angles are varied from $50^\circ$ to $150^\circ$ to study the jet subsonic directivity. Figure 4.1(b) shows the definition used for observer location in the current work. The aircraft during takeoff has a relative velocity component that constitutes a parallel stream surrounding the core jet flow. The parallel stream has a noticeable impact on the noise generation from the jet and should be taken into consideration in studying subsonic jet noise during takeoff. Jets in flight are found to be quieter than the jets in a stationary medium. The following section presents the noise predictions made for the single stream jets at varying co-flow Mach numbers and performs a forward flight analysis for these operating
conditions.

Figure 4.28(a) compares the measured and predicted noise spectra for an unheated jet at $M_j = 0.9$ without co-flow. The experiments were performed on a 2 in convergent nozzle. It is observed that there is a spectral shift between the measured and predicted SPL at lower inlet angles of 90° and 110° but a better match has been obtained at inlet angles of 130° and 150°. The spectra match well up to $St$ of 0.5. Comparison with other experiments [86] shown in figure 4.29 at 90° match well up to $St = 0.3$. Figure 4.28(b) compares the measured and predicted spectra at different inlet angles for a heated jet without co-flow. It can be seen that the spectra match well at $St$ up to 0.4 at all inlet angles. Reasonable agreement between the experiments and predicted values have been observed at different jet operating conditions.

Figure 4.30(a) compares the SPL levels at different inlet angles for unheated and heated jet without co-flow. The rise in SPL values at all frequencies is observed in the heated case and a similar observation is made at all inlet angles as shown by the shift in the OASPL levels in figure 4.30(b).

Figure 4.31(a) compares the SPL values as a function of $St$ at different inlet angles for unheated and heated jets in a parallel stream of $M_{cf} = 0.2$. A similar shift in the SPL levels in the heated jet is observed as in the case without co-flow but there is a higher shift in the levels at intermediate $St$ between 0.1 and 0.3. The OASPL shows a rise in the heated jet with co-flow at all inlet angles as observed in figure 4.30(b).

Forward flight effects on jet noise during takeoff have been studied before experimentally. Von Glahn et al. [94], Cocking and Bryce [95], Bushell [96], Packman et al. [97], Plumblee [93], Tanna and Morris [56] and Cocking [98] carried out some of the earlier experimental studies to study flight effects. The SAE report [55] presented the variation of the flight velocity exponent obtained at both subsonic and supersonic jet exit Mach numbers. This report shows very different values of the exponent as compared to other sources such as Tanna and Morris [56]. To understand this discrepancy between the experiments performed at different facilities, similar calculations have been performed using the present solver for noise calcu-
Figure 4.28. (a) Predicted and experimental [87] ($SMC00 \, D_j = 2 \, in \, nozzle, T_j/T_\infty = 0.835$) 1/3-octave noise spectra plotted against $St$ for convergent nozzle, $M_j = 0.9$ and $TTR = 1.0$, $M_{cf} = 0.0$ at $r/D_j = 72$. (b) Predicted and experimental [86] ($D_j = 1 \, in$, $TTR = 2.6$) 1/3-octave noise spectra plotted against $St$ for convergent nozzle, $M_j = 0.9$ and $TTR = 2.7$, $M_{cf} = 0.0$ at $r/D_j = 72$. 
Figure 4.29. Predicted and experimental [86] \((D_j = 1\ \text{in}, \ TTR = 1.0)\) 1/3-octave noise spectra plotted against \(St\) for convergent nozzle, \(\theta = 90^\circ\), \(M_j = 0.9\) and \(TTR = 1.0\).

lations. The co-flow Mach number varies from 0.0 to 0.28 which represents a true range of flight operating conditions during takeoff.

Figure 4.32(a) shows the variation of the flight coefficient values obtained experimentally for an unheated \(M_j = 0.9\) jet, with tunnel Mach number varying from \(M_t = 0.048 - 0.20\). It is observed that the velocity exponent is nearly equal to 5.0 at lower inlet angles and goes up to 7.5 – 8.0 at larger inlet angles. Figure 4.32(b) shows the variation of flight coefficient reported in SAE ARP876 Report [55] which shows an entirely different trend as reported in other experiments. The coefficient goes to zero at lower inlet angles and increases sharply at higher inlet angles up to 10.0. The coefficient is also a function of jet exit Mach number at higher inlet angles. The method in this paper is based upon data obtained on a ground based engine flight simulation facility. Since full scale engine tests were used for these calculations, the exponent values are different from the model scale wind tunnel tests. Model scale wind tunnel testing has, in general produced greater reduction in level than tests carried out on ground based engine facilities and aircraft. The flight velocity exponent in this work [55] is calculated using equation 4.33.

\[
\Delta OASPL(\theta) = 10\log_{10} \left\{ \left( \frac{V_j}{V_j - V_{cf}} \right)^{m(\theta)} \left( 1 - M_a \cos \phi \right) \right\} \quad (4.33)
\]
(a) 1/3-octave SPL, $TTR = 1.0$ and $2.7$

(b) OASPL vs. $\theta$, $TTR = 1.0$ and $2.7$

Figure 4.30. (a) Numerical 1/3-octave noise spectra for different inlet angles at $r/D_j = 72$ plotted against $St$ for convergent nozzle without parallel stream, $M_j = 0.9$ and $TTR = 1.0$ and $2.7$. (b) Predicted OASPL at different inlet angles for $TTR = 1.0$ and $2.7$ and $r/D_j = 72$. 
Figure 4.31. Convergent nozzle operating at $M_j = 0.9$, $M_{cf} = 0.2$, and $TTR = 1.0$ and 2.7: (a) Numerical 1/3-octave noise spectra for different inlet angles at $r/D_j = 72$ plotted against $St$. (b) Predicted OASPL at different inlet angles at $r/D_j = 72$. 

(a) 1/3-octave SPL, $M_{cf} = 0.2$, $TTR = 1.0$ and 2.7

(b) OASPL vs. $\theta$, $M_{cf} = 0.2$, $TTR = 1.0$ and 2.7
Figure 4.32. Flight velocity coefficient obtained in experiments: (a) Unheated $M_j = 0.9$ jet, tunnel Mach number $M_t = 0.048-0.20$. Data also shown from NGTE wind tunnel experiments. Tanna and Morris [56] and (b) SAE ARP876 Report, 1994 [55].

where $\phi$ is the angle between the direction of aircraft motion and direction of sound propagation, $M_a$ is the acoustic Mach number, $m(\theta)$ is the relative velocity exponent and $\theta$ is the angle between engine inlet axis and the line connecting an aircraft reference point and the observer location. The additional Doppler factor, $(1 - M_a \cos \phi)$ was included in determining the change in the OASPL in this work.
The value of this factor depends on $\phi$. Equation \ref{eq:4.33} reduces to equation \ref{eq:4.34} at $\phi = 90^\circ$.

$$\Delta OASPL(\theta) = 10\log_{10} \left\{ \frac{V_j}{V_j - V_{cf}} \right\}^m$$  \hspace{1cm} (4.34)

Figure \ref{fig:4.33(a)} compares the predicted 1/3-octave noise spectra for an unheated jet with varying co-flow Mach numbers at different inlet angles. It is observed that the SPL lowers at all frequencies with increasing co-flow Mach number. The scatter in the data is due to the shorter sampling time for noise predictions. The finer grid will resolve the spectra at higher frequencies and longer sampling time will result in smoother spectra. Figure \ref{fig:4.33(b)} shows OASPL for different co-flow Mach numbers for unheated jet as a function of inlet angles. The levels get lower with increasing co-flow Mach number at all inlet angles; the difference is more at higher inlet angles while it is less at lower inlet angles. It should be noted that though the absolute agreement deteriorates at higher Strouhal numbers, it is expected that relative changes due to the effect of forward flight will be captured.

The co-flow analysis can be performed by relating the reduction in the OASPL to the logarithmic value of the ratio of the jet exit speed and its difference with the parallel stream speed. The least squares fit between the reduction in noise levels and $10\log_{10} \left\{ \frac{V_j}{V_j - V_{cf}} \right\}$ gives the slope $m$ which relates the reduction in OASPL to the relative jet speed by equation \ref{eq:4.34}.

Figure \ref{fig:4.34} shows the least squares fit between $\Delta OASPL$ and relative jet speed at different inlet angles. The flight velocity exponent $m$ can then be plotted as a function of nozzle inlet angle as shown in figure \ref{fig:4.35(a)}. Figure \ref{fig:4.35(b)} shows the variation of $m$ obtained experimentally by Viswanathan and Czech \cite{54} for a similar nozzle for a range of jet operating conditions. A better match has been obtained with the values of flight velocity exponent with recent experiments\cite{54} as compared to previous measurements\cite{56, 55} as shown in figures \ref{fig:4.32(a)} and \ref{fig:4.32(b)}.

There is clear dependence on inlet angle of the value of the flight velocity exponent $m$. Both numerical and experimental values show a similar trend of variation of $m$ with inlet angle. It increases gradually from 3.0 at lower polar angles $\sim$ 50 to 105°
Figure 4.33. (a) Numerical prediction of 1/3-octave noise spectra plotted against $St$ for convergent nozzle, $M_j = 0.9$ and $TTR = 1.0$, for different co-flow Mach numbers $M_{cf} = 0.0$, 0.12, 0.2 and 0.28 at $r/D_j = 72$. (b) Comparison of OASPL at different inlet angles for single stream nozzle, $M_j = 0.9$ and $TTR = 1.0$, for different co-flow Mach numbers $M_{cf} = 0.0$ to 0.28 at $r/D_j = 72$. 

(a) Predicted SPL, $M_{cf} = 0.0-0.28$, $TTR = 1.0$

(b) Predicted OASPL, $M_{cf} = 0.0-0.28$, $TTR = 1.0$
Figure 4.34. Least square fit between reduction in the OASPL and different co-flow speeds for $\theta = 150^\circ$ to $50^\circ$ for single stream nozzle, $M_j = 0.9$, $TTR = 1.0$, $M_{cf} = 0.0$ to $0.28$.

(a) Predicted flight velocity exponent
(b) Experimental[54] flight velocity exponent

Figure 4.35. (a) Numerical variation of flight velocity exponent $m$ for different inlet angles, $M_j = 0.9$, $TTR = 1.0$. (b) Experimental variation of the flight velocity exponent for different inlet angles given in Viswanathan and Czech[54]. $m$ to about 5.5 at $\sim 110$ to $150^\circ$. The measurements report higher values (up to 8.0) at higher inlet angles as compared to the numerical data. Jet spectra scaling can be performed using the flight velocity exponent information as shown in figure 4.36. The scaled 1/3-octave SPL is calculated as $SPL + m \times 10\log_{10} \left\{ \frac{V_j}{V_j - V_{cf}} \right\}$. A good match has been found between spectra at $M_{cf} = 0.0$ and $M_{cf} = 0.12$. The scaling using just one parameter $m$ seems to work well across all the frequencies. This
scaling can be used to predict the noise spectra for other co-flow Mach numbers.

4.8 Summary

This chapter has presented flow and noise prediction results for single stream convergent jets at different operating conditions. It has been found that both jet temperature and parallel flow affects the flow and hence the noise characteristics of the jet. The heated jets radiate more noise than the cold jets operating at the same jet exit Mach number due to the rise in the absolute jet exit velocity. The same heating effects are found in jets with and without co-flow. The jets in flight are quieter than the corresponding jet without any parallel stream. The forward flight analysis shows that the scaling laws can be used to predict the spectra at different co-flow flight conditions. The numerical results agree well with wind
tunnel experiments. The aircraft flight tests have shown different results for the flight effects which might be due to the contribution from other noise sources more prominent in co-flow conditions. The flight effects are greater in the peak noise direction as shown by a higher value of the flight velocity exponent. The turbulence properties on the lip line show a different behavior for jets in flight. The effect of reducing the shear between the jet and surrounding flow affects the turbulent flow characteristics such as eddy convection velocity and decay of turbulence.

The next chapter presents flow simulations and noise predictions made for high-subsonic dual-stream jets in flight. A flight effect analysis has been performed for dual-stream jets and comparisons have been made with the single stream jet findings.
This chapter presents the flow and noise simulation results performed for dual-stream nozzle operating at high subsonic jet exit conditions. A Boeing-designed convergent nozzle with an area ratio of $A_s/A_p = 3.0$ is used for the flow simulations. The flow properties of the dual-stream jets are studied and compared with the single stream jets. The statistical analysis of turbulence is performed and the results are compared with the values obtained for single stream jet calculations. The flight effects are analyzed for dual-stream jets using different co-flow Mach numbers. Comparisons are made with the single stream jet findings. A simplified pylon structure is incorporated in the dual-stream nozzle to simulate the installation effects on jet noise. The flow simulations with the pylon are compared with the baseline nozzle. The following sections present the flow and noise calculations followed by the forward flight analysis for the dual-stream nozzle. The installation effects are then presented.

5.1 Introduction

Dual-stream jet exits are found in turbofan engines used in present day commercial aircraft. In a typical dual-stream exhaust, the heated core is surrounded by the unheated fan stream, which gives the dual-stream configuration. The details of the
dual-stream flow field and the difference between single and dual-stream jets were given in Chapter 1. The dual-stream or coaxial nozzle are set up to simulate the noise for actual high-bypass engine operating conditions. The coaxial nozzle used in the present work is a Boeing designed convergent nozzle, with an area ratio of $A_s/A_p = 3.0$ where the primary nozzle extends beyond the secondary nozzle. This configuration is representative of large turbofan engines in current service.

The core nozzle exit diameter $D_{pj}$ is 2.45 in and that of the fan nozzle $D_{sj}$ is 6.00 in. The primary nozzle area is 4.714 sq in and secondary nozzle area is 14.142 sq in. The equivalent diameter, taken for normalizing the axial and radial distances, is calculated from the volumetric flow rate of the core and fan nozzle, and is found to be $V_{mix} \pi D_e^2/4 = V_p A_p + V_s A_s = 4.9$ in. The heated core jet velocity is $V_p = 1347$ ft/s (408 m/s), the fan nozzle velocity is $V_s = 1049$ ft/s (320 m/s), and the mixed jet velocity can be calculated using the mass conservation as:

$$V_{mix} = \frac{V_p A_p \rho_p + V_s A_s \rho_s}{A_p \rho_p + A_s \rho_s}$$

(5.1)

For heated core jet averaged jet velocity is equal to $V_{mix} = 1093$ ft/s. The test matrix for the jet operating conditions is shown in Table 5.1. These test conditions are chosen for the current work as the experimental data are available at operating conditions. Although the jet exit Mach number for the secondary nozzle is higher as compared to the primary jet exit Mach number, the jet speed is much higher in primary nozzle as heated flow passes through the primary nozzle.

Cases I and II have unheated core and fan flows while cases III and IV have heated core and unheated fan flow which is more typical of aircraft engine conditions. The total temperature ratio is chosen to give the core flow temperature of approximately 600 K. The ambient conditions are 298 K temperature and 101,000 Pa pressure.

<table>
<thead>
<tr>
<th>Case</th>
<th>$M_{pj}$</th>
<th>$M_{sj}$</th>
<th>$TTT_p$</th>
<th>$TTT_s$</th>
<th>$M_{cf}$</th>
<th>bypass ratio ($\dot{m}_s/\dot{m}_p$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>0.85</td>
<td>0.95</td>
<td>1.0</td>
<td>1.0</td>
<td>0.0</td>
<td>3.3</td>
</tr>
<tr>
<td>II</td>
<td>0.85</td>
<td>0.95</td>
<td>1.0</td>
<td>1.0</td>
<td>0.2</td>
<td>3.3</td>
</tr>
<tr>
<td>III</td>
<td>0.85</td>
<td>0.95</td>
<td>2.26</td>
<td>1.0</td>
<td>0.0</td>
<td>4.55</td>
</tr>
<tr>
<td>IV</td>
<td>0.85</td>
<td>0.95</td>
<td>2.26</td>
<td>1.0</td>
<td>0.2</td>
<td>4.55</td>
</tr>
</tbody>
</table>

Table 5.1. Dual-stream jet operating conditions, $D_{pj} = 2.45$ in, $D_{sj} = 6.00$ in.
The flow simulation and noise prediction results are presented in the following sections for the cases listed above. The co-flow analysis is of particular interest in this case. The heated core is surrounded by a high speed fan flow which shields the core jet plume from the ambient conditions. This characteristic of dual-stream jets is different from that of single stream nozzles. The observer location is $122D_e$ with nozzle inlet angles varying from $50^\circ$ to $150^\circ$.

### 5.2 Dual-stream Nozzle Case Setup

The dual-stream nozzle simulation is set up in a similar manner as the single stream nozzle, though there are few important differences between the two cases. In the dual-stream nozzle, there are two shear layers that need to be resolved accurately. Therefore, grid clustering is required near both the nozzle exits. Different total inlet conditions are specified at the two nozzle inlets as both nozzles are operating at different jet exit conditions. The flow calculations are performed at high subsonic Mach numbers in the primary and secondary nozzles ($M_{pj} \sim 0.85$, $M_{sj} \sim 0.95$) with the total pressure ratios in the primary nozzle of $NPR_p = 1.6$ and in the secondary nozzle of $NPR_s = 1.8$. The total number of CPU cores used in this calculation is 100 and the computation time to achieve developed turbulent flow is 22 days.

Figure 5.1(a) shows the multi-block grid inside the dual-stream nozzle with grid clustering near the walls. The coaxial nozzle is a convergent nozzle with area ratio of $A_s/A_p = 3.0$ where the primary nozzle extends beyond the secondary nozzle. The grid resolution is selected on the basis of the shortest resolvable wavelength. The grid stretching ratio is maintained at 1.05 - 1.1 until $30D_e$ downstream of the nozzle exit. Figure 5.1(b) shows the mesh surrounding the nozzles. Figure 5.2 shows the computational domain and grid slices extracted inside and outside the nozzles at fixed axial locations. The domain extends up to $70D_{pj}$ in the axial direction and up to $20D_{pj}$ in the radial direction. Figure 5.3(a) shows the grid clustering in the primary and secondary shear layer regions to help capture the flow transition and generate unsteadiness. Figure 5.3(b) shows the far-field observer location with
Figure 5.1. Multi-block grid topology inside and outside the dual-stream nozzle with $D_{pj} = 2.45 \text{ in} \ (0.0622 \text{ m})$ and $D_{sj} = 6.00 \text{ in} \ (0.15 \text{ m})$. (a) Cylindrical mesh surrounding the H-type mesh in the center to remove centerline singularity, (b) Cylindrical mesh surrounding the dual-stream nozzle.

Figure 5.2. Computational domain for dual-stream jet simulation: Black lines show the axial and radial extent of the computational domain. The primary nozzle geometry is shown in blue surrounded by secondary nozzle in red. Inset (left) shows a slice extracted inside the nozzles. A cylindrical mesh surrounds the H-type mesh in the center to remove centerline singularity. Inset (right) shows the grid at a slice at a downstream location outside the nozzle.
139

(a) Multi-block mesh inside dual-stream nozzle  (b) Observer location with respect to core nozzle exit

Figure 5.3. [a] Grid clustering to resolve primary and secondary shear layers in the dual-stream nozzle.  [b] Observer location specified in terms of polar distance $r$ and nozzle inlet angle $\theta$ with respect to the core nozzle exit.

respect to the primary nozzle exit in terms of polar distance $r$ and inlet angle $\theta$.

5.3 Flow Simulation Results

The dual-stream nozzle calculations are performed for jet operating conditions typical of a commercial aircraft engine\cite{35} with $NPR_p = 1.6$, $TTR_p = 1.0$ and 2.26, $NPR_s = 1.8$ and $TTR_s = 1.0$, $M_{cf} = 0.0$ and 0.2. The time averaged axial velocity contours for the unheated core nozzle are shown in figures 5.4(a) and 5.4(b). The primary jet potential core extends beyond the secondary jet potential core region. Two shear layers are formed in the initial mixing region which merge further downstream. The potential core elongates in the case of co-flow as shown in figure 5.4(b). Figures 5.4(c), 5.4(d) show the instantaneous pressure contours for the same test conditions with $M_{cf} = 0.0$ and 0.2. It can be seen that the jet plume width and jet spreading rate reduce in case of the presence of parallel stream $M_{cf} = 0.2$. The parallel flow effects the primary nozzle flow even in the presence of the high speed fan flow shielding the core flow.

Figures 5.5(a), 5.5(b) and 5.5(c) show the mean axial velocity contours for $TTR_p$,
Mean axial velocity contours, $TTR_p = 1.0$, $M_{cf} = 0.0$

Instantaneous pressure perturbation contours:

$c) M_{cf} = 0$, $d) M_{cf} = 0.2$.

Figure 5.4. Mean axial velocity and temperature contours for dual-stream nozzle $NPR_p = 1.6$, $TTR_p = 1.0$, $NPR_s = 1.8$, $TTR_s = 1.0$, with parallel stream Mach numbers, $M_{cf} = 0.0$ and 0.2: $a) M_{cf} = 0$, $b) M_{cf} = 0.2$. Instantaneous pressure perturbation contours: $c) M_{cf} = 0$, $d) M_{cf} = 0.2$.

= 2.26 and co-flow Mach number of $M_{cf} = 0.0$, 0.2 and 0.28 respectively. It can be seen that the potential core length increases with a parallel flow surrounding the fan nozzle in heated core jet as well. Figures 5.5(d), 5.5(e) and 5.5(f) show the pressure perturbation contours for $TTR_p = 2.26$ and co-flow Mach number of $M_{cf} = 0.0$, 0.2 and 0.28 respectively. The unsteady eddies shown by the pressure perturbation values become elongated in the axial direction in the presence of parallel flow and the elongation increases further for $M_{cf} = 0.28$.

Figures 5.6(a) and 5.6(b) show the mean temperature contours without and with co-flow respectively. The elongation in the thermal potential core region is visible with the parallel flow. It can be seen that the thermal potential length is shorter as compared to the dynamic potential core. Figures 5.6(c) and 5.6(d) show the turbulent kinetic energy contours (equation 4.12) for $M_{cf} = 0.0$ and 0.2 respectively. $TKE$ represents the energy in the turbulent eddies and it is observed that the most energetic eddies lie in two regions in the dual-stream flow: at the end of primary jet potential core and in the lip line of the fan stream between $3-5D_{pj}$ downstream.
Figure 5.5. Mean axial velocity and instantaneous pressure contours for different coflow Mach numbers for dual-stream jet operating at $NPR_{p} = 1.6$, $TTR_{p} = 2.26$, $NPR_{s} = 1.8$, $TTR_{s} = 1.0$, $M_{cf} = 0.0$, 0.2, 0.28.
of the primary nozzle exit. These energy-containing eddies act as prominent noise sources in the jet flow. Figure 5.6(e) shows a snapshot of the instantaneous density contours for a heated core jet with $M_{cf} = 0.2$. The lower density heated core flow is surrounded by the unheated higher density fan flow.

Figures 5.7(a)-5.7(b) show the variation of the axial velocity at the nozzle exit on the walls of the core and fan nozzles respectively for the cold core jet. It can be seen that the boundary layer is similar with and without co-flow at the core nozzle.
boundary layers on core nozzle walls, $TTR_p = 1.0$, $M_{cf} = 0.0$ and $0.2$

(b) Boundary layers on fan nozzle walls, $TTR_p = 1.0$, $M_{cf} = 0.0$ and $0.2$

(c) Boundary layers on core nozzle walls, $TTR_p = 2.26$, $M_{cf} = 0.0$ and $0.2$

(d) Boundary layers on fan nozzle walls, $TTR_p = 2.26$, $M_{cf} = 0.0$ and $0.2$

Figure 5.7. Boundary layers development on the walls of the (a), (c) core and (b), (d) fan nozzles for dual-stream convergent nozzle operating at $NPR_p = 1.6$, $TTR_p = 1.0$ and $2.26$, $NPR_s = 1.8$, $TTR_s = 1.0$, $M_{cf} = 0.0$ and $0.2$.

Exit while the boundary layer develops to $M_{cf} = 0.2$ on the fan nozzle wall for the co-flow case. The boundary layer thickness on the nozzle walls determines the potential core length of the jet flow. A similar observation is made for the boundary layers on the nozzle walls for the heated core flow as shown in figures 5.7(c), 5.7(d).

The mean axial velocity is normalized with the core jet exit velocity in both the cases. Hence, the ratio $<u_1>/U_{pj}$ is greater than 1 in the fan nozzle in the unheated core case while it is less than 1 in the heated core case.
The displacement thickness \( \delta^* = \int_0^\infty \left( 1 - \frac{\rho(y)u(y)}{\rho_o u_o} \right) dy \) is calculated for \( M_{cf} = 0.0 \) and 0.2 on both sides of the core and fan nozzle walls and its variation is shown in figures 5.8(a) and 5.8(b). \( \rho_o \) and \( u_o \) are the reference values away from the boundary layer, for example, they are equal to free-stream values for the outer surface of the nozzle wall and equal to the jet centerline values at the nozzle exit for the inner surface of the nozzle wall. It is observed that the boundary layer remains nearly the same on both inner and outer walls of the primary nozzle at both co-flow Mach numbers. Therefore, the parallel stream does not have any significant impact on the boundary layer development on the unheated core jet nozzle walls. The displacement thickness on the inner wall of the fan nozzle decreases with increasing co-flow Mach number while \( \delta^* \) increases on the outer wall with co-flow Mach number as expected.

Figures 5.8(c)-5.8(d) show the \( \delta^* \) values on the inner and outer walls of the core and fan nozzles for the heated core case at \( M_{cf} = 0.0 \) and 0.2. The displacement thickness inside the nozzles is nearly constant at all co-flow Mach numbers. The \( \delta^* \) on the outer surface of the fan nozzle increases with co-flow Mach number as expected while it decreases with the co-flow Mach number on the outer wall of the core nozzle. It suggests that the parallel flow pushes the fan stream towards the core jet flow thus decreasing the boundary thickness on the outer wall of the core nozzle.

Figure 5.9(a) shows the variation of the centerline mean axial velocity and centerline and lip line turbulence intensity profiles with and without a parallel stream with Mach number of \( M_{cf} = 0.2 \) at \( TTR_p = 1.0 \). The fan flow reduces the growth of the primary shear layer and hence elongates the primary potential core. Jet mixing noise is produced primarily at the end of the potential core. Hence, the noise sources are pushed further downstream in the dual-stream nozzle as compared to a similar single stream jet. It is observed that the parallel stream surrounding the fan flow further elongates the potential core length even though the high speed fan stream provides a shielding effect on the hot core flow. The turbulence intensity levels reduce with parallel flow along the centerline and primary and secondary shear layers. This shows the effect of the parallel stream on both the core and fan flows. Figure 5.9(b) shows the centerline properties for a heated core jet with
(a) $\delta^*$ on inner surfaces of nozzle walls, $TTR_p = 1.0$
(b) $\delta^*$ on outer surfaces of nozzle walls, $TTR_p = 1.0$
(c) $\delta^*$ on inner surfaces of nozzle walls, $TTR_p = 2.26$
(d) $\delta^*$ on outer surfaces of nozzle walls, $TTR_p = 2.26$

Figure 5.8. Displacement thickness ($\delta^*$) values on (a),(c) inner and (b),(d) outer surfaces of the core and fan nozzles for dual-stream jet $NPR_p = 1.6$, $TTR_p = 1.0$ and 2.26, $NPR_s = 1.8$, $TTR_s = 1.0$, with parallel stream Mach numbers, $M_{cf} = 0.0$ and 0.2

$M_{cf} = 0.0$ and 0.2. Observations similar to the unheated core jet are made in the heated core jet as well. The potential core length elongates with the co-flow Mach number. The jet potential core shortens in the heated jet as compared to the unheated jet and thus the peak turbulence intensity values are shifted in the upstream direction. The fan nozzle lip line shows a higher value of turbulence intensity near the primary jet exit. It should be noted that the fan nozzle exit lies upstream of the core nozzle exit and therefore the turbulent eddies have developed in the fan lip line at the axial location of primary jet exit.

Figure 5.9(c) shows the centerline variation of the mean axial velocity for unheated
Figure 5.9. Variation of centerline mean axial velocity and centerline and shear layers turbulence intensity values along the jet axis for jet with co-flow $M_{cf} = 0.0$ and 0.2, $NPR_p = 1.6$, $NPR_s = 1.8$, $TTR_s = 1.0$: (a) $TTR_p = 1.0$, (b) $TTR_p = 2.26$, (c) Centerline $< U >$ values, $TTR_p = 1.0$ and $2.26$, (d) Centerline $< T >$ values, $TTR_p = 2.26$ and 2.26.

Variation of mean centerline temperature values along the jet axis for jet operating at $NPR_p = 1.6$, $TTR_p = 2.26$, $NPR_s = 1.8$, $TTR_s = 1.0$ with co-flow $M_{cf} = 0.0$ and 0.2 and heated core jets with and without co-flow. The axial location is normalized with respect to the dynamic potential core length. Good agreement has been found between the normalized centerline velocity profiles for all cases. Figure 5.9(d) shows the variation of the centerline relative mean temperature profile along the jet axis with and without co-flow. The axial location is normalized with respect to the thermal potential core length. Both normalized profiles collapse with each other well. The thermal variation along the jet centerline shows a similar trend as
observed in single stream jets.

The development of the unsteady flow in the axial direction downstream of the nozzle exit can be seen in the vorticity slices plotted at different axial locations as shown in figures 5.10(a)–5.10(d). It can be seen that the flow is steady and smooth at the core nozzle exit (0.015$D_{pj}$) and starts to transition to turbulent flow at 1$D_{pj}$. The shear layer becomes turbulent as the jet develops downstream, and this turbulent mixing results in a fully turbulent jet between 5$D_{pj}$ − 10$D_{pj}$. The flow at the fan nozzle shear is already in the transition mode at 0.015$D_{pj}$ as the fan nozzle exit is upstream of the core nozzle exit. The two streams merge downstream and a uniform mixed flow is obtained after 5$D_{pj}$. A similar observation can be made for $M_{cf} = 0.2$. Eddies elongate in the axial direction with increasing co-flow as observed in the pressure perturbation contours.

Figure 5.11(a) shows the radial variation of the relative temperature at different axial locations downstream of the nozzle exit for the heated core jet with $M_{cf} = 0.2$. It is observed that the jet has achieved a self-similar thermal profile at all axial locations $x/D_{pj} > 2.0$. Radial profiles within 2.0 < $x/D_{pj} < 5.0$ fall upstream of the end of the thermal potential core while radial profiles between 6.0 < $x/D_{pj}$ < 12.0 fall beyond the end of the thermal potential core. All the radial profiles collapse very well in the two regions. Figure 5.11(b) show the radial variation of the mean axial velocity at different axial locations upstream and downstream of the end of the dynamic potential core. Good collapse has been obtained in the radial profiles downstream of the dynamic potential core region where the primary and secondary jets have mixed forming a uniform mixed flow region. Upstream of the end of the potential core, a jump is observed in the velocity profile near $r/r_{0.5U_{CL}} \sim 0.65$ which corresponds to the centerline region of the fan stream. This jump smears out downstream of the nozzle exit where the two streams have merged together.

Table 5.2 summarizes the unheated and heated primary jet ($T_{pj} = 578$ K) flow characteristics with and without co-flow. It can be seen that the potential core length decreases with heated core flow as compared to the unheated jet. A similar trend was observed for the single stream jet. The ratio of the thermal and dynamic
Figure 5.10. Vorticity magnitude contours plotted for dual-stream jet operating at \( M_{pj} = 0.85, M_{sj} = 0.95, TTR_s = 1.0, M_{cf} = 0.0 \) and 0.2. Slices extracted at different axial locations (0.015 < \( x/D_{pj} \) < 10) downstream of the nozzle exit to show the transition from laminar to turbulent flow in the shear layer. Both streams merge to form an uniform mixing region downstream of the nozzle. (a) \( TTR_p = 1.0, M_{cf} = 0.0 \), (b) \( TTR_p = 1.0, M_{cf} = 0.2 \), (c) \( TTR_p = 2.26, M_{cf} = 0.0 \), (d) \( TTR_p = 2.26, M_{cf} = 0.2 \).
Figure 5.11. Radial profiles of the relative temperature (a) and mean axial velocity (b) showing jet similarity inside and outside potential core region, $TTR_p = 2.26$, $M_{cf} = 0.2$. Radial distance normalized with half temperature radius and half velocity radius respectively.
Table 5.2. Comparison of jet flow characteristics for dual-stream jet operating at $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 1.0$ and 2.26, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, and $M_{cf} = 0.0$ and 0.2.

<table>
<thead>
<tr>
<th>$TTR_p$</th>
<th>$M_p$</th>
<th>$M_{pa}$</th>
<th>$M_{cf}$</th>
<th>$x_{CU}/D_{pj}$</th>
<th>$x_{CT}/D_{pj}$</th>
<th>$x_{CT}/x_{CU}$</th>
<th>$(u'<em>{CL})</em>{max}/U_j$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>0.85</td>
<td>0.85</td>
<td>0.0</td>
<td>10.5</td>
<td>-</td>
<td>-</td>
<td>0.13</td>
</tr>
<tr>
<td>2.26</td>
<td>0.85</td>
<td>1.20</td>
<td>0.0</td>
<td>6.05</td>
<td>4.7</td>
<td>0.75</td>
<td>0.14</td>
</tr>
<tr>
<td>1.0</td>
<td>0.85</td>
<td>0.85</td>
<td>0.2</td>
<td>12.91</td>
<td>-</td>
<td>-</td>
<td>0.10</td>
</tr>
<tr>
<td>2.26</td>
<td>0.85</td>
<td>1.20</td>
<td>0.2</td>
<td>8.41</td>
<td>6.77</td>
<td>0.81</td>
<td>0.12</td>
</tr>
</tbody>
</table>

Potential core lengths is equal to 0.75 which is smaller than the value obtained in the single stream jet (0.90). It reflects that the effect of the heated flow surrounded by the cold flow enhances the thermal mixing as compared to the single stream case where the heated jet is surrounded by the ambient stationary flow. The turbulence intensity levels rise with jet exit temperature as expected. The dynamic potential core length increases with co-flow in both heated and unheated core jets. The maximum turbulence intensity values decrease with co-flow in both heated and unheated core jets, a similar trend was observed in single stream jets.

Figure 5.12 shows the iso-surfaces of the $Q$-criterion plotted for the dual-stream flow with unheated core flow and without a parallel flow. The iso-surfaces show the turbulent flow eddies in the jet plume. The figure also shows the pressure perturbation contours on the acoustic data surface on which data sampling is performed for the noise calculations.

Figures 5.13(a) and 5.13(b) show the iso-surfaces of $Q$-criterion for the heated core jet at $M_{cf} = 0.0$ and 0.2. It can be seen that in the jet in the parallel stream the turbulent eddies are elongated in the axial direction and the jet plume becomes narrower. A similar observation was made in single stream jets as well.
Figure 5.12. Iso-surfaces of $Q$ criterion ($Q = \pm 50^5 \text{s}^{-2}$) for dual-stream nozzle operating at $NPR_p = 1.6$, $TTR_p = 1.0$, $NPR_s = 1.8$, $TTR_s = 1.0$, $M_{cf} = 0$. Pressure perturbations shown on the enclosing acoustic surface.

(a) $Q$-criterion, $M_{cf} = 0.0$  
(b) $Q$-criterion, $M_{cf} = 0.2$

Figure 5.13. Iso-surfaces of $Q$ criterion ($Q = \pm 10^6 \text{s}^{-2}$) for dual-stream nozzle operating at $NPR_p = 1.6$, $TTR_p = 2.26$, $NPR_s = 1.8$, $TTR_s = 1.0$: (a) $M_{cf} = 0.0$, (b) $M_{cf} = 0.0$ and 0.2

5.4 Statistical Analysis of Turbulence for Jets in Flight

A statistical analysis of turbulence is performed for the dual-stream jet to study the behavior of the turbulent motion in the primary and secondary shear layers and in the radial direction at the end of the potential core. Two point correlations are
calculated with the probes fixed at $x/D_{pj} = 2$ and 3 for the primary and secondary shear layers respectively. The probe locations are shown in figure 5.14. The radial locations are $y/D_{pj} = 0.5$ and 1.0.

Figures 5.15(a)-5.15(c) show the second order two-point axial velocity correlation values on the primary lip line for the unheated core jet and heated core jets at $M_{cf} = 0.0$ and 0.2. Similar correlation coefficients are obtained in unheated and heated jets without co-flow. The peak correlation values fall more rapidly with probe separation distance in the case of the heated jet with co-flow as seen in figure 5.15(c). Figures 5.15(d)-5.15(f) show the fourth order two-point correlation values on the primary lip line for the unheated core jet and heated core jets at $M_{cf} = 0.0$ and 0.2. The fourth-order correlation coefficients decay more rapidly with probe separation distance as was seen in single stream jets. The eddies decorrelate more rapidly in the heated jet with co-flow. The decay is slowest in the heated jet without co-flow. The envelope joining the peaks of the correlation functions gives the eddy convection velocity on the primary nozzle lip line.

Figures 5.16(a)-5.16(c) show the variation of the second order two-point axial velocity correlation coefficients on the secondary lip line with the reference probe at $x/D_{pj} = 3.0$ for the unheated core jet and heated core jets at $M_{cf} = 0.0$ and 0.2. The correlation graphs are similar to the values obtained in the primary shear. Among the three cases, the decay is slowest in the heated case without co-flow. The decay of the peaks is slower on the secondary lip line as compared to the
Figure 5.15. Dual-stream nozzle operating at $M_p = 0.85$, $M_s = 0.95$, $TTR_s = 1.0$, and $M_{cf} = 0.0$ and $TTR_p = 1.0$ and 2.26: Second order two point correlation coefficients plotted along the primary shear layer for (a) $TTR_p = 1.0$, $M_{cf} = 0.0$. (b) $TTR_p = 2.26$, $M_{cf} = 0.0$. (c) $TTR_p = 2.26$, $M_{cf} = 0.2$. Fourth order two point correlation coefficients plotted along the primary shear layer for (d) $TTR_p = 1.0$, $M_{cf} = 0.0$. (e) $TTR_p = 2.26$, $M_{cf} = 0.0$. (f) $TTR_p = 2.26$, $M_{cf} = 0.2$. $\xi$ is the separation distance between two probes. Correlations plotted in the shear layer $y/D_{pj} = 0.5$ at $x/D_{pj} = 2.0$. 
primary lip line values. Figures 5.16(d)-5.16(f) show the variation of the fourth order two-point correlation values on the fan lip line for the three cases. The decay is faster in the fourth order formulation as observed in primary lip line values. It should be noted that the data for correlation analysis is extracted at constant radial distances while the centerline of the shear layers is tilted towards the jet axis due to the converging nature of the fan nozzle.

Figures 5.17(a)-5.17(c) show comparisons of the spatial variation of the second order and fourth order cross-correlation coefficients calculated from equation 4.25 and from the relation $c_{1111} = r_{11}^2$ (See Morris and Zaman [91]) on the primary lip line. Both formulations for the fourth order correlation coefficients agree very well for all the three cases. A similar variation of the correlation coefficients is observed in the three cases except that negative values of the second order correlation coefficients are observed at larger separation distances in the heated jet with co-flow.

Figures 5.17(d)-5.17(f) show the variation of time delay for maximum correlation coefficient with separation distance for the unheated core jet, the heated core jet without and with co-flow respectively. The separation distance increases with time delay for the maximum correlation value in all three cases. Both second order and fourth order correlation values are plotted. Linear curves can be fitted to the values, the slope of which gives the convection velocity. The slopes of both second order and fourth order coefficients are nearly the same in all three cases. It can be seen that the slope decreases in the case of the heated core jet as compared to the unheated core jet.

Figures 5.18(a)-5.18(c) show comparisons of the spatial variation of the second order and fourth order cross-correlation coefficients calculated from equation 4.25 and from the relation $c_{1111} = r_{11}^2$ (See Morris and Zaman [91]) on the secondary nozzle lip line. Both formulations for the fourth order correlation coefficients agree very well for all the three cases. A similar variation of the correlation coefficients is observed in all the three cases. Figures 5.18(d)-5.18(f) show the variation of the time delay for maximum correlation coefficient with separation distance for the unheated core jet, the heated core jet without and with co-flow on the secondary nozzle lip line. The separation distance increases with time delay for maximum correlation value in all three cases. Both second order and fourth order correlation
(a) Secondary Shear: \( r_{11} \), \( TTR_p = 1.0 \), \( M_{cf} = 0.0 \)

(b) Secondary Shear: \( r_{11} \), \( TTR_p = 2.26 \), \( M_{cf} = 0.0 \)

(c) Secondary Shear: \( r_{11} \), \( TTR_p = 2.26 \), \( M_{cf} = 0.2 \)

Figure 5.16. Dual-stream nozzle operating at \( M_p = 0.85 \), \( M_s = 0.95 \), \( TTR_s = 1.0 \), and \( M_{cf} = 0.0 \) and \( TTR_p = 1.0 \) and 2.26: Second order two point correlation coefficients plotted along the secondary shear layer for (a) \( TTR_p = 1.0 \), \( M_{cf} = 0.0 \), (b) \( TTR_p = 2.26 \), \( M_{cf} = 0.0 \), (c) \( TTR_p = 2.26 \), \( M_{cf} = 0.2 \). Fourth order two point correlation coefficients plotted along the secondary shear layer for (d) \( TTR_p = 1.0 \), \( M_{cf} = 0.0 \), (e) \( TTR_p = 2.26 \), \( M_{cf} = 0.0 \), (f) \( TTR_p = 2.26 \), \( M_{cf} = 0.2 \). \( \xi \) is the separation distance between two probes. Correlations plotted in the shear layer \( y/D_{pj} = 1.0 \) at \( x/D_{pj} = 3.0 \).
(a) Primary Shear: $TTR_p = 1.0$, $M_{cf} = 0.0$  
(b) Primary Shear: $TTR_p = 2.26$, $M_{cf} = 0.0$  
(c) Primary Shear: $TTR_p = 2.26$, $M_{cf} = 0.2$  

(d) Primary Shear: $TTR_p = 1.0$, $M_{cf} = 0.0$  
(e) Primary Shear: $TTR_p = 2.26$, $M_{cf} = 0.0$  
(f) Primary Shear: $TTR_p = 2.26$, $M_{cf} = 0.2$  

**Figure 5.17.** Dual-stream nozzle operating at $M_p = 0.85$, $M_s = 0.95$, $TTR_s = 1.0$, and $TTR_p = 1.0$ and 2.26:  
(a) Correlation coefficients at zero time delay for unheated core jet.  
(b) Correlation coefficients at zero time delay for heated core jet.  
(c) Correlation coefficients at zero time delay for heated core jet with $M_{cf} = 0.2$.  
(d) Variation of time delay for maximum cross correlation with separation distance for unheated core jet.  
(e) Variation of time delay for maximum cross correlation with separation distance for heated core jet.  
(f) Variation of time delay for maximum cross correlation with separation distance for heated core jet with $M_{cf} = 0.2$.  

$\xi$ is the separation distance between two probes. Correlations plotted along the primary lip line $y/D_{pj} = 0.5$ at $x/D_{pj} = 2.0$. 

156
values are plotted. Linear curves can be fitted to the values, the slope of which gives the convection velocity. It can be seen that the slope decreases in the case of the heated core jet as compared to unheated core jet. The slopes for the fourth order coefficients are nearly the same as compared to the second order coefficients. The overall trend of the plots is similar to the properties observed along the primary nozzle lip line.

Figures 5.19(a)-5.19(c) show the radial variation of the flatness factor defined by equation 4.31 at \( x/D_{pj} = 5.0 \) for different jet conditions. It is observed that the flatness value remains nearly constant across the radial direction in the range of 3.0-4.0 for all three cases, also seen in single stream jet measurements [91]. The values are lowest on the lip line i.e., \( r - 0.5D_{pj} = 0 \), which is consistent with the measurements. The information about flatness factor \( T_1(x) \) is required when the quasi-normal hypothesis is not used for relating the fourth order and second order auto-correlation coefficients. Figures 5.19(d)-5.19(f) show the variation of phase velocity calculated using equation 4.32 for the unheated core jet and heated core jet with and without co-flow. Logarithmic curve fits [91, 92] are also plotted for comparison. It is observed that the numerical data is closer to the curve fits in the heated core jet cases as compared to the unheated jet. A similar observation was made for the single stream jets. This scatter in the data is primarily due to wiggles in the phase delay at different separation distances. A longer sampling time should smooth the phase delay and hence provide a more uniform convective phase velocity.

Table 5.3 lists the various length and time scales calculated for the unheated and heated core jet with co-flow on the primary and secondary jet lip lines. The scales are calculated in a similar manner as outlined in Chapter 4. The value of time scale \( L_{\tau,r_1} U_j/D_j \) increases in the heated core jet as compared to the unheated jet on both primary and secondary lip lines. But, its value decreases in the presence of the co-flow on both lip lines. The time scale based on cross-correlation \( L_{\tau,r_1} U_j/D_j \) increases in the heated jet and also with co-flow on the primary jet lip line. But its value decreases in the co-flow case along the secondary jet lip line. The length scales based on second order and fourth order coefficients show a similar trend on both primary and secondary jet lip lines. The values increase for the heated core
(a) Secondary Shear: $TTR_p = 1.0$, $M_{cf} = 0.0$

(b) Secondary Shear: $TTR_p = 2.26$, $M_{cf} = 0.0$

(c) Secondary Shear: $TTR_p = 2.26$, $M_{cf} = 0.2$

(d) Secondary Shear: $TTR_p = 1.0$, $M_{cf} = 0.0$

(e) Secondary Shear: $TTR_p = 2.26$, $M_{cf} = 0.0$

(f) Secondary Shear: $TTR_p = 2.26$, $M_{cf} = 0.2$

Figure 5.18. Dual-stream, nozzle operating at $M_p = 0.85$, $M_s = 0.95$, $TTR_p = 1.0$, and $TTR_s = 1.0$ and 2.26: (a) Correlation coefficients at zero time delay for unheated core jet. (b) Correlation coefficients at zero time delay for heated core jet. (c) Variation of time delay for maximum cross correlation with separation distance for heated core jet. (d) Variation of time delay for maximum cross correlation with separation distance for heated core jet with $M_{cf} = 0.2$. (e) Correlation with separation distance between two probes. Correlation plotted along the secondary lip line $y/D_{pj} = 1.0$ at $x/D_{pj} = 3.0$. $\xi$ is the separation distance between two probes.
(a) Flatness value in the radial distance, $TTR_p = 1.0$, $M_{cf} = 0.0$

(b) Flatness value in the radial distance, $TTR_p = 2.26$, $M_{cf} = 0.0$

(c) Flatness value in the radial distance, $TTR_p = 2.26$, $M_{cf} = 0.2$

(d) Primary Shear: $TTR_p = 1.0$, $M_{cf} = 0$

(e) Primary Shear: $TTR_p = 2.26$, $M_{cf} = 0$

(f) Primary Shear: $TTR_p = 2.26$, $M_{cf} = 0.2$

Figure 5.19. Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 2.26$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$. (a) Flatness ($T_1(x)$) parameter along the radial direction at $x/D_{pj} = 5.0$ for $M_{cf} = 0.0$, $TTR_p = 1.0$. (b) Flatness ($T_1(x)$) parameter along the radial direction at $x/D_{pj} = 5.0$ for $M_{cf} = 0.2$, $TTR_p = 2.26$. (c) Flatness ($T_1(x)$) parameter along the radial direction at $x/D_{pj} = 5.0$ for $M_{cf} = 0.2$, $TTR_p = 2.26$. Variation of phase velocity as a function of Strouhal number (d) $TTR_p = 1.0$, $M_{cf} = 0.0$, (e) $TTR_p = 2.26$, $M_{cf} = 0.0$, (f) $TTR_p = 2.26$, $M_{cf} = 0.2$. 
Table 5.3. Predicted length and time scales, and convection speed of the turbulent structures of dual-stream jets ($TTR_p = 1.0$ and 2.26) operating with and without co-flow. Calculated at $x/D_{pj} = 2.0$ on the core lip line and at $x/D_{pj} = 3.0$ on the fan lip line based on cross-correlation of the axial velocity fluctuations.

<table>
<thead>
<tr>
<th>$TTR_p$</th>
<th>$M_{cf}$</th>
<th>$L_{r,11}U_j/D_j$</th>
<th>$L_{r,x_{11}}U_j/D_j$</th>
<th>$L_{x,x_{11}}/D_j$</th>
<th>$U_c/U_j$</th>
</tr>
</thead>
<tbody>
<tr>
<td>core lip line</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.0</td>
<td>0.0</td>
<td>0.4514</td>
<td>0.8587</td>
<td>0.5279</td>
<td>0.4071</td>
</tr>
<tr>
<td>2.26</td>
<td>0.0</td>
<td>0.9780</td>
<td>0.9905</td>
<td>0.6313</td>
<td>0.4527</td>
</tr>
<tr>
<td>2.26</td>
<td>0.2</td>
<td>0.6362</td>
<td>1.1124</td>
<td>0.3973</td>
<td>0.2916</td>
</tr>
<tr>
<td>fan lip line</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.0</td>
<td>0.0</td>
<td>0.4589</td>
<td>0.7771</td>
<td>0.4461</td>
<td>0.4663</td>
</tr>
<tr>
<td>2.26</td>
<td>0.0</td>
<td>0.8604</td>
<td>2.3352</td>
<td>0.6970</td>
<td>0.6312</td>
</tr>
<tr>
<td>2.26</td>
<td>0.2</td>
<td>0.6378</td>
<td>0.9350</td>
<td>0.5101</td>
<td>0.3796</td>
</tr>
</tbody>
</table>

jet as compared to the unheated jet but go down in the heated jet with co-flow. The eddy convection velocity decreases in the heated core jet as compared to the unheated core jet but again increases in the presence of the parallel stream in the heated core jet case. A similar observation is made on both primary and secondary jet lip lines. Single stream jets show different behavior in the presence of a parallel stream and the convection velocity further decreases in the co-flow cases as shown in table 4.5.

5.5 Noise Predictions and Forward Flight Analysis

Noise calculations are performed for the dual-stream jet using a similar tapered FW-H surface surrounding the fan nozzle and extending up to $30D_{pj}$ downstream of the nozzle exit. The surface extends to approximately $1.5D_{pj}$ at the nozzle exit in the radial direction. A typical FW-H surface for a dual-stream jet is shown in figure 5.20.

Numerical and experimental (Ref.: Viswanathan and Czech[54]) noise spectra calculated at different observer angles are shown in figure 5.21 for a heated core jet without co-flow. The observer is located at $122D_e$ from the core nozzle exit. The
numerical and experimental values match well at lower frequencies and a reasonable agreement is obtained between the numerical and measured OASPL values. The spectra fall off at higher \( St \) due to grid resolution constraints as observed in the single stream jets.

Figure 5.22(a) compares the 1/3-octave SPL at different nozzle inlet angles with and without co-flow for the unheated core jet. It can be seen that the parallel stream suppresses the noise levels at all angles; a similar trend being observed with the single stream jets. Figure 5.22(b) compares the OASPL as a function of nozzle inlet angles with and without co-flow. The noise levels reduce by about 6 dB in the presence of co-flow at all inlet angles, which is consistent with the trend observed in the single stream nozzles. These calculations are made for an unheated core flow.

Figure 5.23(a) compares the 1/3-octave SPL for the heated core jet with and without co-flow at different inlet angles. The radial observer distance is \( 122D_e \). It can be seen that the noise levels are lower with co-flow at lower frequencies at all inlet angles but the two spectra collapse at higher frequencies. This might be due to the grid resolution limit at the higher frequencies. Figure 5.23(b) compares the OASPL at different inlet angles for the heated core jet with and without co-flow. A reduction in noise levels in the presence of co-flow is observed at all angles, although the reduction is less as compared to the unheated core jet case.

To perform the forward flight analysis, a procedure similar to the one used for
Figure 5.21. Experimental and numerical 1/3-octave noise spectra compared at different nozzle inlet angles at $122D_e$ for dual-stream nozzle with $NPR_p = 1.6$, $TTR_p = 2.26$, $NPR_s = 1.8$, $TTR_s = 1.0$, $M_{cf} = 0.0$. Equivalent jet diameter $D_e = 4.9$ in. Strouhal number calculated using primary jet parameters.

single stream jets is followed. Since only two data points corresponding to two co-flow Mach numbers are available, very reliable estimates of the flight velocity exponent cannot be performed, but an initial estimate has been made here. In dual-stream nozzles, more than one value is available to be used as the jet characteristic velocity. Two values are used here to study the forward flight effect. The first value is the primary jet exit velocity denoted by $V_{pj}$ and the second value is the jet mixed velocity denoted by $V_{mix}$. Equation 5.1 is used to calculate $V_{mix}$ for the heated and unheated core jet cases. The reduction in OASPL is plotted against the co-flow Mach number for the unheated core jet as shown in figures 5.24(a) and 5.24(b). Least square fits can be drawn through these curves for different inlet angles. $V_{pj}$ is used as the reference velocity in these figures. The slopes of these curve fits give the flight velocity exponents as a function of inlet angle as shown in figure 5.24(c).

Figures 5.24(d) and 5.24(e) plot the reduction in OASPL in presence of co-flow
(a) SPL vs. $St$ for $TTR_p = 1.0$, $M_{cf} = 0.0$ and $0.2$, $r/D_e = 122$.

Figure 5.22. Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 1.0$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, $M_{cf} = 0.0$ and $0.2$. Equivalent jet diameter $D_e = 4.9$ in. Strouhal number calculated using primary jet flow parameters. (a) $\frac{1}{3}$-octave SPL compared at different nozzle inlet angles at $122D_e$. (b) Variation of overall noise levels (OASPL) with different nozzle inlet angles at $122D_e$ with $M_{cf} = 0.0$ and $0.2$. 

(b) OASPL vs. $\theta$ for $TTR_p = 1.0$, $M_{cf} = 0.0$ and $0.2$, $r/D_e = 122$. 

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure522.png}
\caption{Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 1.0$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, $M_{cf} = 0.0$ and $0.2$. Equivalent jet diameter $D_e = 4.9$ in. Strouhal number calculated using primary jet flow parameters. (a) $\frac{1}{3}$-octave SPL compared at different nozzle inlet angles at $122D_e$. (b) Variation of overall noise levels (OASPL) with different nozzle inlet angles at $122D_e$ with $M_{cf} = 0.0$ and $0.2$.}
\end{figure}
(a) SPL vs. St for TTR$_p$ = 2.26, $M_{cf}$ = 0.0 and 0.2, $r/D_e$ = 122.

(b) OASPL vs. $\theta$ for TTR$_p$ = 2.26, $M_{cf}$ = 0.0 and 0.2, $r/D_e$ = 122.

Figure 5.23. Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, TTR$_p$ = 1.0, $M_s = 0.95$, $NPR_s = 1.8$, TTR$_s$ = 1.0, $M_{cf} = 0.0$ ans 0.2. Equivalent jet diameter $D_e$ = 4.9 in. Strouhal number calculated using primary jet flow parameters. (a) 1/3-octave SPL compared at different nozzle inlet angles at 122$D_e$. (b) Variation of overall noise levels (OASPL) with different nozzle inlet angles at 122$D_e$ with $M_{cf} = 0.0$ and 0.2.
Figure 5.24. Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 1.0$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, $V_{pj} = 291 \text{ m/s}$, $V_{mix} = 315.0 \text{ m/s}$. (a), (b) Least squares fit curves for $\Delta OASPL$ vs. $M_{cf}$ at different inlet angles using $V_{ref} = V_{pj}$. (c) Variation of flight velocity exponent with nozzle inlet angles using $V_{ref} = V_{pj}$. (d), (e) Least squares fit curves for $\Delta OASPL$ vs. $M_{cf}$ at different inlet angles using $V_{ref} = V_{mix}$. (f) Variation of flight velocity exponent with nozzle inlet angles using $V_{ref} = V_{mix}$.
as a function of relative velocity where $V_{\text{mix}}$ is used as the reference velocity. The least squares fit drawn at different inlet angles can be used to calculate the flight velocity exponent as shown in figure 5.24(f). It can be seen that the use of $V_{\text{mix}}$ shifts up the flight velocity exponent curve plotted as a function of inlet angle. In this case $V_{\text{mix}} = 315 \text{ m/s}$ which is greater than the value of $V_{\text{pj}} = 291 \text{ m/s}$ which increases the slope of the curve fits. It can be seen that with the available data, the gradual increment in the flight velocity exponent with inlet angle is not observed as is the case with the single stream jet. Instead, the velocity exponent lowers at $110^\circ - 120^\circ$ before increasing again in the peak noise direction. However this is likely due to there being insufficient data at different flight velocities.

Figures 5.25(a) and 5.25(b) show the reduction in OASPL in the presence of co-flow for the heated core jet at different inlet angles. $V_{\text{pj}}$ is used as the reference velocity in this case. The flight velocity exponent plotted against the nozzle inlet angle is shown in figure 5.25(c). For lower inlet angles, the velocity exponent values are similar to those obtained for the unheated core jet (figure 5.24(c)) but the exponent obtained is ($\sim 3.0-4.0$) at the higher inlet angles ($\theta > 100^\circ$) for the heated jet as compared to the higher values of $\sim 3.0-5.0$ for the unheated core jet. Figures 5.25(d) and 5.25(e) show the reduction in OASPL in the presence of co-flow for the heated core jet at different inlet angles. $V_{\text{mix}}$ is used as the reference velocity in this case. The flight velocity exponent plotted against the nozzle inlet angle is shown in figure 5.25(f). In this case, $V_{\text{mix}}$ (334 m/s) is lower than $V_{\text{pj}}$ (408 m/s) and hence shifts the velocity exponent curve lower. Comparison with the experiments would give better insight in making a more appropriate choice for the reference jet velocity. Jet noise spectra scaling can be performed using the flight velocity exponent as shown in the single stream jet cases in Chapter 4. Figure 5.26(a) shows the scaled spectra at $\theta = 130^\circ$ and $150^\circ$ for the unheated core jet with $V_{\text{pj}}$ as the reference velocity. It can be seen that both spectra collapse well at all $St$. Figure 5.26(b) shows the scaled spectra at $\theta = 130^\circ$ and $150^\circ$ for the unheated core jet with $V_{\text{mix}}$ as reference velocity. The spectra scale well at all frequencies in this case as well. It should be noted that the scaling will be independent of the choice of reference jet velocity as the scaling term $m \ast 10\log_{10}(V_{\text{ref}}/(V_{\text{ref}} - V_{\text{cf}}))$ has same value irrespective of the choice of $V_{\text{ref}}$. 
Figure 5.25. Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 2.26$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, $V_{pj} = 408.0$ m/s, $V_{mix} = 334.0$ m/s. (a) Least squares fit curves for $\Delta OASPL$ vs. $M_{cf}$ at different inlet angles using $V_{ref} = V_{pj}$. (c) Variation of flight velocity exponent with nozzle inlet angles using $V_{ref} = V_{pj}$. (d) Least squares fit curves for $\Delta OASPL$ vs. $M_{cf}$ at different inlet angles using $V_{ref} = V_{mix}$. (d) Variation of flight velocity exponent with nozzle inlet angles using $V_{ref} = V_{mix}$.
168

(a) Scaling using flight velocity exponent, $V_{ref} = V_{pj}$

(b) Scaling using flight velocity exponent, $V_{ref} = V_{mix}$

Figure 5.26. Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 1.0$, $M_s = 0.95$, $NPR_s = 1.8$, $TTR_s = 1.0$, $V_{pj} = 291.0$ m/s, $V_{mix} = 315.0$ m/s. [a] Jet noise spectra scaling using $V_{pj}$, [b] Jet noise spectra scaling using $V_{mix}$. $r/D_e = 122$. Black circles: $M_{cf} = 0$, Red triangles: $M_{cf} = 0.2$

Figure 5.27(a) shows the scaled noise spectra for the heated core jet at $\theta = 130^\circ$ and $150^\circ$ with $V_{pj}$ as reference velocity. Figure 5.27(b) shows the scaled noise spectra for the heated core jet at $\theta = 130^\circ$ and $150^\circ$ with $V_{mix}$ as reference velocity. It can be seen that the two spectra collapse well at all $St$ except at very low frequencies at both inlet angles.

5.6 Dual-Stream jet with Installation Effects

In realistic engine configurations, it is necessary to include the installation devices and other aircraft components that interact with the jet plume and thus can affect
the resultant jet noise in terms of either directivity or magnitude or both. More
details about the influence of the installation devices such as a pylon on
turbofan flow and noise characteristics are given in Chapter 1 Section 1.1.3. In the present
research, a simplified version of the actual pylon is used to simulate installation
effects on the dual-stream jet flow and noise. The simplified version is used to
reduce the grid complexity but it retains the important features such as the percent-
age of fan nozzle area blockage and the overall size. The following sub-sections
present the details of the grid generation, case setup and flow simulation results
for the nozzle with pylon structure.

Figure 5.27. Dual-stream nozzle with $M_p = 0.85$, $NPR_p = 1.6$, $TTR_p = 2.26$, $M_s =$
0.95, $NPR_s = 1.8$, $TTR_s = 1.0$, $V_{pj} = 408.0 ~m/s$, $V_{mix} = 334.0 ~m/s$. (a) Jet noise
spectra scaling using $V_{pj}$. (b) Jet noise spectra scaling using $V_{mix}$. $r/D_e = 122$. Black
circles: $M_{cf} = 0$, Red triangles: $M_{cf} = 0.2$
5.6.1 Case Setup for Nozzle with Pylon

The pylon structure is derived from realistic pylon-nozzle geometries. Figure 5.28(a) shows the dual-stream jet with a pylon. The pylon structure extends up to $2D_{pj}$ downstream of the core nozzle which is representative of an actual configuration. The pylon blocks $\sim 8.5\%$ of the cross-sectional area of the fan nozzle. There are in total 36 blocks in the computational domain. Non-matching boundaries are used to refine the grid near the pylon walls which allows the generation of a computational mesh of reasonable size. Figure 5.28(b) shows the one half of the computational domain with the pylon. The dark red lines represent the pylon structure. Figure 5.29 shows the grid clustering near the primary and secondary shear layers and on the pylon walls. The black lines show the multi-block grid topology. A viscous no-slip wall condition is imposed on the pylon walls.

The mesh size has nearly 11 million grid points for the coaxial jet with a pylon. Approximately 120 processors are used in the simulation and one month’s computational time is required to complete the flow simulation and noise sampling on the acoustic data surface. Total flow conditions are used at the nozzle inlets to achieve the required pressure and velocity at the nozzle exit. The jet operating conditions are similar to those used for the baseline dual-stream nozzle. The core nozzle is unheated and a parallel stream with $M_{cf} = 0.1$ is used to simulate the co-flow. A forward flight case is required to study the impact of the pylon on the resultant noise.

The fan-nozzle diameter is increased to keep the resultant cross-sectional area the same as the baseline nozzle. This will isolate the effect of the pylon on the jet noise. Otherwise, by keeping the same nozzle geometry in both calculations, the resultant jet exit area will not remain the same in both geometries, which will in turn affect the noise levels. Since the pylon breaks the azimuthal symmetry of the jet, a variation of the noise in the azimuthal plane is expected due to the presence of the pylon. The noise directivity should remain similar in the polar direction. The next section presents the flow simulation results obtained for the pylon case and comparisons are made with the baseline nozzle.
Figure 5.28. Grid generation for dual-stream nozzle with pylon $D_{pj} = 2.45\, in$ and $D_{sj} = 6.1\, in$ [a] Dual-stream nozzle showing simplified pylon structure, [b] One half computational domain showing block boundaries and pylon structure (solid red lines)

5.6.2 Flow Simulation Results

The flow simulations are performed on the multi-block grid described in the previous section. The instantaneous Mach contours and the vorticity contours are shown along different planes in figures 5.30(a)-5.30(d). It can be seen that the presence of the pylon distorts the axisymmetric nozzle geometry which is different
from the axisymmetric nature seen in baseline nozzle (see figure 5.4(b)) and thus affects the mean flow field. Similar observations are made in the vorticity contours in and out of the pylon plane. The out of pylon plane vorticity contours still retain a symmetric nature as shown in figure 5.30(d) but figure 5.30(c) shows the eddies developing at the end of the pylon and higher vorticity levels are obtained due to the flow swirl at the pylon end. These features will affect the resultant noise calculated in the far-field in an installed engine configuration. Figures 5.30(a) and 5.30(b) show snapshots of the instantaneous Mach number contours in and out of pylon plane which also clearly show the effect of the pylon intruding the flow in its plane. Figure 5.32 shows the vorticity contours near the pylon end and it can be seen that the flow has higher instability in the shear layers developed due to the pylon walls and the jet tends to spread more in the radial direction. A thick boundary layer is developed on the pylon wall as evident in figure 5.31(a). The out of the pylon plane mean profile shown in figure 5.31(b) still retains the symmetric nature of the jet plume. Figure 5.33 shows the mean axial velocity contours at different axial locations. It can be seen that the pylon distorts the mean axial velocity profiles especially until the end of pylon.
Figure 5.30. Dual-stream with pylon jet operating at $M_p = 0.85$, $M_s = 0.95$, $TTR_p = 1.0$, $TTR_s = 1.0$ and $M_{cf} = 0.12$. Instantaneous Mach contours (a) in pylon plane, (b) perpendicular to the pylon plane. Vorticity contours (c) in pylon plane, (d) perpendicular to the pylon plane.
(a) Mean axial velocity $< u >$, in the plane of the pylon

(b) Mean axial velocity $< u >$, out of the pylon plane

**Figure 5.31.** Jet operating at $M_p = 0.85$, $M_s = 0.95$, $TTR_p = 1.0$, $TTR_s = 1.0$ and $M_{cf} = 0.12$: Mean axial velocity contours (a) in the pylon plane, (b) out of the pylon plane.

**Figure 5.32.** Vorticity contours near the pylon end showing flow circulation.
Figure 5.33. Mean axial velocity slices at different axial locations showing asymmetric nature of the jet plume.

5.7 Summary

This chapter presents the flow and noise prediction results for dual-stream convergent nozzles at different operating conditions. The turbulence properties in the primary and secondary nozzle lip lines show a different behavior for jets in flight. The effect of reducing the shear between the jet and surrounding flow affects the turbulent flow characteristics such as the eddy convection velocity and the decay of correlation of the turbulence. It has been found that both jet temperature and parallel flow affects the flow and hence noise characteristics of the jet. The heated jets radiate more noise than the cold jets operating at same jet exit Mach number due to the rise in the absolute jet exit velocity. The same heating effects are found in jets with and without co-flow. The jets in flight are quieter than the corresponding jet without any parallel stream. Two reference velocities have been used to perform the forward flight analysis and to calculate the flight velocity exponent. The choice of reference velocity shifts the exponent values but does not affect the spectra scaling. The forward flight analysis shows that the scaling laws can be used to predict the spectra at different co-flow flight conditions.
The installation effects are studied by including a simplified pylon structure with the baseline dual-stream nozzle. The flow field shows that the pylon breaks the axisymmetric nature of the nozzle and effects the flow field mostly in the pylon plane. This intrusion in the flow field will effect the resultant noise prediction for the nozzle with pylon.
Conclusions and Future Work

In the current research, jet noise simulations have been performed on high-subsonic single and dual-stream nozzles. The flight effects on the jet noise for both single and dual-stream nozzles have been studied and analyzed. The engine installation effects were studied with a simplified pylon structure installed on the dual-stream nozzle. A modified DES approach was used to perform the flow simulations and solutions to the Ffowcs Williams and Hawkings equation were used for far-field noise calculations. This chapter presents the conclusions made from the present study related to the numerical approach and the jet noise results. Finally, suggestions for the extension of the current work are given.

6.1 Conclusions

The present research used a hybrid RANS-LES approach for jet flow simulations. The simulations were performed for single and dual-stream nozzle at high-subsonic Mach numbers which were representative of the actual aircraft flying conditions. The turbulence flow simulations first resolved the near-field turbulent noise sources. The unsteady data was sampled on an acoustic data surface which was then used to predict the far-field noise. An acoustic data surface was assumed to enclose all major noise sources. The present numerical methodology is tailored for accurate
aeroacoustic applications under the constraint of limited computer resources. The following sub-sections present the conclusions made from the different numerical strategies used in the present work and the jet noise simulations for single and dual-stream nozzles.

6.1.1 Numerical Methods

The major components of the EAGLEJet and PSFWH solvers have been discussed in detail in Chapter 2. Here, the major issues associated with the implementation of these methods and the reasons for these strategic choices are discussed:

- The 3D compressible unsteady Navier-Stokes equations with one-equation Spalart-Allmaras turbulence model have been used to perform the turbulent flow simulations. In the hybrid RANS-LES approach, the eddy viscosity was turned off in the DES region to avoid excessive damping of the small turbulent eddies. For numerical stability of the turbulence model, a limit was set on the maximum value of the eddy viscosity.

- A multi-block structured grid topology was used to generate meshes for the more complex nozzle geometries. The structured grid provides the flexibility of achieving higher grid resolution near the nozzle walls as required and can be coarsened away from the walls. A fine mesh has been generated to resolve the finite nozzle lip thickness which is important to trigger the instability in the shear layer. The centerline singularity was removed by using a H-type Cartesian grid in the center of the nozzle. The grid singularity generated due to the 3D multi-block topology where several block boundaries share a common grid point was solved using the characteristic interface conditions for data communication.

- Steady local time marching was performed to accelerate the development of the jet flow. The standard fourth order Runge-Kutta scheme with modified coefficients has been used for the local time stepping.

- Dual-time stepping was used to simulate the time-accurate unsteady flow
from the steady state flow field. Sub-iterations have been performed using local time marching, while the physical time was used for time-accurate marching. This method accelerates the convergence of the flow-field to a statistically stable state.

- Implicit residual smoothing was performed with local time marching to modify the residual which further resulted in accelerated convergence rate. The coefficients used in the implicit residual smoothing were adjusted to obtain a numerically stable solution.

- The spatial discretization was performed using seven-point DRP scheme which minimizes the dispersion of the acoustic waves. Lower order stencils were used near the domain boundaries.

- An efficient parallel communication method has been used in this work to provide good scalability for large computations. The solver used MPI for data communication across the CPUs. Two level communicator handles were created: within the grid block and across the two block boundaries. The solver efficiently distributed the load to the processors to minimize the processor waiting time and communication overhead time. The output solution files were written by individual processors which alleviates the need for one processor to hold all the data for file writing.

- The characteristic interface boundary conditions were used for data communication at the block interfaces. The solver included the capability to deal with different grid orientations at the interface. A higher order interpolation method was implemented to communicate the data at non-matching block boundaries.

- An Immersed Boundary Method based on the Brinkman penalization method was implemented in the solver to incorporate small geometric changes without modifying the grid. For example, this method has been recently successfully used to simulate chevrons on a military style supersonic jet\cite{70}.

- The noise predictions were made using the Ffowcs-Williams and Hawkings Formulation 1A equation. The moving source term has been incorporated to
accurately calculate the noise for co-flow cases.

- Radiation and wall boundary conditions were implemented to achieve accurate computational boundary requirements for jet flow simulations. The Riemann boundary conditions were also implemented to accurately simulate the inlet boundaries for the parallel stream.

All numerical methods were implemented in the EAGLEJet solver to perform jet flow simulations. The PSFWH code was modified to work with dimensional input data. All the numerical methods were validated against several benchmark cases. Good agreement was obtained with the analytic solutions for all the cases as shown in Chapter 3.

6.1.2 Jet Noise Simulations

The jet flow and noise calculations have been performed for high-subsonic unheated and heated single and dual-stream nozzles for varying co-flow Mach numbers. Several observations have been made from the flow and noise results for both nozzle geometries. The following sub-sections present the major conclusions drawn from the jet noise calculations.

6.1.2.1 Single Stream Jets

A Boeing-designed convergent nozzle has been used in the present research for noise simulations of single stream jets. The jet operating conditions are: $M_j = 0.9$, $M_{cf} = 0.0$ to 0.28, $TTR = 1.0$ and 2.7. The simulations were performed with multiple numerical strategies before evaluation of the final numerical methods to achieve the best results under the given computational constraints. Different grid resolutions and acoustic data surface locations were explored to study their effects on the jet noise and appropriate selections were made to obtain the best possible results. Several issues that greatly affect the jet noise results have been found as:

- The grid resolution in several regions of the computational domain for the nozzle plays an important role in resolving spectra at a given range of fre-
quencies. The grid needs to be resolved in both axial and radial directions near the nozzle exit to accurately capture the high frequency content of the noise spectra. It has been found that the grid stretching in the axial direction should be $\sim 1.05-1.06$ up to $25D_j$ downstream of the nozzle exit to accurately capture all the noise sources. A fine grid is required in the radial direction as well up to $2-3D_j$ as the high energy containing eddies should be well resolved in the shear layer for accurate noise calculations. The three degree grid resolution is found to be sufficient in the azimuthal direction. Different levels of grid refinement are required in heated jets with co-flow where the actual jet exit velocity is much higher as compared to the unheated jets and a parallel stream adds to the boundary layer growth on the outer surface of the nozzle walls.

- The numerical methods used to discretize the partial differential equations have numerical errors in them. A practical CFD solver requires an artificial dissipation mechanism to ensure that the solution remains stable but the external dissipation should not modify the physics of the flow field. In the current work, the effect of different dissipation switches based on pressure, density and pressure-density gradients has been studied for the jet flow cases and it was found that the pressure only switch provided the least dissipation to the turbulent fluctuations. It should be noted that the dissipation method should be able to keep the solution stable.

- The location of the acoustic data surface used to collect the data for noise calculation is important in the jet noise studies. The surface should be able to enclose all major noise generating sources and the resolution on the surface should be fine enough to capture the smaller wavelengths. Two parameters have been varied to find the optimum location for the surface: the axial end of the surface and the radial distance of the surface sleeve at the nozzle exit. It has been found that the axial distance should be large enough so that the passage of the turbulent eddies through the outflow disk can be minimized. Moving the closure distance downstream will improve the low frequency content. The radial distance should be such that it encloses the jet plume containing the large eddies but should not be very far from the
nozzle wall as the spectral resolution will be compromised.

A computational mesh with 5.5 million grid points has been generated for single stream convergent nozzle. There are 14 blocks in total and the grid is significantly refined near the nozzle exit to resolve the high frequency noise sources up to $St \sim 2.0$. The unheated and heated jet flow and noise simulations were performed for the jet operating conditions mentioned earlier. The flight effects on the jet noise have been assessed for the single stream jets and the following conclusions are made from the flow and noise results:

- The grid refinement study performed for the single stream jet with 3.5 and 5.0 million grid points mesh shows that the finer grid did improve the spectral resolution at high frequencies at all nozzle inlet angles. The OASPL obtained with the finer mesh had better agreement with the experiments.

- The entrainment in the jet flow was assessed by calculating the mean flow rate along the axial distance for the heated jet and the flow entrainment was observed to be similar to other numerical work\cite{22}.

- The vorticity slices extracted at different axial locations for different co-flow Mach numbers showed the delay in transition to turbulence as the co-flow Mach number increases. The centerline turbulence intensity value decays with increasing co-flow Mach number which also suggests that the parallel stream diminishes the high energy containing eddies in the jet. Higher co-flow velocity reduces the shear layer gradient and hence reduces the jet shear layer instability. An elongation of the potential core was observed with increasing co-flow Mach number which is consistent with experimental observations. The iso-surfaces of $Q$-criterion also showed the elongation of eddies in the downstream direction and the narrowing of jet plume with increasing co-flow Mach number. The centerline and radial variation of the mean axial velocity profiles matched well with experiments for different co-flow Mach numbers.

- The value of the potential core length was under-predicted in the numerical simulations as compared to some experiments but, there has been disagreement between the different experiments on the value of the potential core
length. The potential core length depends on the initial evolution of the jet, jet contraction ratio and the boundary layer development on the nozzle walls, which might be different in different experimental settings. The boundary layer on the outer and inner surface of the nozzle walls has been analyzed and the displacement thickness values were calculated at different co-flow Mach numbers. The growth of the boundary layer with increasing co-flow Mach number was evident on the outer surface of the nozzle while the boundary growth on the inner surface was not affected by the parallel stream. The displacement thickness value on the outer wall increased with co-flow Mach number while it remains almost constant on the inner surface.

- The radial variation of the mean axial velocity profile and relative temperature profiles show two self-similar regions in the jet flow: upstream and downstream of the end of the potential core. The radial profiles collapse well in these regions. This shows that the self-similar behavior of the developed jet is achieved in the present simulations. A similar observation is made for the axial turbulence intensity levels plotted against a radial similarity coordinate, which is consistent with the experiments.

- The two-point correlation analysis provides an insight into the development and decay of the turbulence in the shear layer as a function of co-flow Mach number. It was observed that the convection speed deceased in heated jets and with co-flow in comparison with the unheated jet without co-flow. The time scale increased in the heated jet and increased further in the presence of co-flow. A higher value means that the eddy decay rate is slower in flows with a parallel stream which is consistent with the lower levels of peak turbulence in high co-flow cases. The length scales based on second order correlation decreased in the heated jet as compared to unheated jet but a higher value was obtained for a jet with parallel stream. The opposite behavior was obtained with the fourth order length scale which reduced in the heated jet with co-flow as compared to cold and heated jets without co-flow.

- The noise spectra at different inlet angles matched reasonably well with the experiments for no co-flow unheated and heated cases. The agreement was
good for lower frequencies up to $St \sim 0.4$ but fell off at higher $St$. Better grid resolution would improve the spectral content at higher frequencies. Also, a longer sampling time would provide smoother spectra. The effect of heat on the jet noise was assessed by running a jet simulation at $TTR = 2.7$. The levels rose at all inlet angles for the heated jet as the jet exhaust velocity is much higher for the heated jet as compared to the unheated jet which elevate the noise levels at all frequencies. A similar effect of heating was observed in the presence of a parallel-stream.

- The noise spectra obtained for different co-flow Mach numbers showed that lower noise levels are obtained as the co-flow Mach number increases. This trend is consistent with the experiments and other numerical studies. Higher reduction in OASPL was observed at higher nozzle inlet angles as compared to the lower angles. The calculation of the flight velocity exponent as a function of inlet angle showed the clear dependence of the flight effect on the inlet angle. The effects are greater at higher inlet angles i.e., near the peak noise direction as compared to the lower angles where the noise is mainly governed by the fine scale turbulence. The exponent trend is consistent with the recent experiments although there are discrepancies with the other experiments. The flight velocity exponent increases gradually from 3.0 at lower polar angles $\sim 50$ to $105^o$ to about 5.5 at $\sim 110$ to $150^o$. The measurements report higher values (up to 8.0) at higher inlet angles as compared to the numerical data. Jet spectra scaling can be performed using the flight velocity exponent information. A good match has been found between spectra at different co-flow Mach numbers. The scaling using just one parameter $m$ works well across all frequencies. This scaling can be used to predict the noise spectra for other co-flow Mach numbers.

6.1.2.2 Dual-Stream Jets and Installation Effects

A Boeing-designed dual-stream nozzle with $A_s/A_p = 3.0$ has used in the present work to study the noise characteristics and flight effects on dual-stream jet noise. The methodology used for dual-stream jet calculations is similar to the single
stream jet calculations. The flow characteristics of the dual-stream jet flow are
different from the single stream nozzle because of the presence of high-speed un-
heated flow surrounding the heated core flow. The jet operating conditions are
listed in table 5.1. Both unheated and heated core flows have been studied. The
following observations and conclusions are made for the flow and noise character-
istics of a dual-stream jet:

• The presence of the fan-flow shielded the core flow and increased the jet exit
area and hence reduced the resultant noise levels as compared to a similar
single stream jet configuration. An elongation of potential core has been
observed for dual-stream jets compared to the single stream jet due to the
surrounding high speed flow. The presence of co-flow further increased the
potential core length which suggests that the surrounding flow still impacts
the core flow characteristics even in the presence of the fan-flow. The two
streams are separate up to $3D_{pj}$ downstream after which the two streams
merge and form a uniform turbulent flow similar to a single stream jet. Jet
mixing noise is produced primarily at the end of the potential core. Hence,
the noise sources are pushed further downstream in the dual-stream nozzle
as compared to a similar single stream jet. The turbulence intensity levels
reduce with parallel flow along the centerline and the primary and secondary
shear layers. Similar observations have been made for the heated core jet.
The jet potential core shortens in the heated jet as compared to the unheated
jet and thus the peak turbulence intensity values are shifted in the upstream
direction. The fan nozzle lip line showed a higher value of turbulence intensity
near primary jet exit.

• The boundary layer profile remained nearly the same on both inner and
outer walls of the primary nozzle at both co-flow Mach numbers for the
unheated core jet. Therefore, the parallel stream did not significantly im-
pact the boundary layer growth on the unheated core jet nozzle walls. The
displacement thickness on the inner wall of the fan nozzle decreased with
increasing co-flow Mach number while it increased on the outer wall with
co-flow Mach number as expected. In case of the heated core nozzle, the
displacement thickness inside the nozzles was nearly constant at all co-flow
Mach numbers. The displacement thickness on the outer surface of the fan nozzle increased with co-flow Mach number as expected while it decreased with the co-flow Mach number on the outer wall of the core nozzle. This suggests that the parallel flow pushes the fan stream towards the core jet flow thus decreasing the boundary thickness on the outer wall of the core nozzle.

- The radial relative temperature profiles showed a self-similar thermal profile at all axial locations $x/D_{pj} > 2.0$. Radial profiles within $2.0 < x/D_{pj} < 5.0$ fall upstream of the end of the thermal potential core while radial profiles between $6.0 < x/D_{pj} < 12.0$ fall beyond the end of the thermal potential core. All the radial profiles collapse very well in the two regions. The mean axial velocity profiles, plotted against the normalized radial distance, also showed good collapse downstream of the dynamic potential core region. It should be noted that downstream of the primary potential core, the primary and secondary jets have already mixed forming a uniform mixed flow region. Upstream of the end of the potential core, a jump was recorded in the radial velocity profile whose location corresponds to the centerline region of fan stream.

- The ratio of the thermal and dynamic potential core lengths was smaller than the value obtained in the single stream jet. This reflects that the effect of the heated flow surrounded by the cold flow enhanced the thermal mixing as compared to the single stream case where the heated jet was surrounded by the ambient stationary flow. The turbulence intensity levels rose with jet exit temperature as expected. The dynamic potential core length increased with co-flow in both heated and unheated core jets. The maximum turbulence intensity values decrease in the presence of co-flow for both heated and unheated core jets; a similar trend was observed in the single stream jets.

- Two-point correlations in the primary and secondary shear layers help in determining the motion of the turbulent eddies in the two shear regions present in the dual-stream jet. It has been observed that the eddies decorrelate more rapidly in the primary shear layer in the heated jet with co-flow. The decay
was slowest in the heated jet without co-flow. In the secondary shear layer, the decay was slowest in the heated case without co-flow. The decay of the peaks was slower on the secondary lip line as compared to the primary lip line values. The convection velocities on the core lip line were lower as compared to the values on fan lip line for both the cases. The convection velocity decreased in heated jets as compared to the unheated core jet while it increased in the presence of a parallel stream in both primary and secondary shear layers. This behavior is different from the observations made for the single stream jets.

- The noise spectra calculations performed at different nozzle inlet angles match well with the experiments at lower frequencies but the spectra fell at higher frequencies. A similar observation was made for single stream jets. In the presence of co-flow, the noise levels decrease at all frequencies and at all inlet angles which is consistent with the observations made for a single stream jet. The heated core jet produced higher noise levels as compared to the unheated core jet which is consistent with the observations made for single stream jets. The difference in the OASPL for different co-flow Mach numbers is reduced in case of the heated core jet where the heated jet dominates the noise sources.

- The flight effects have been studied for both unheated and heated core jets and different observations have been made. For dual-stream jets, two reference velocities were used: the primary jet exit velocity and the mixed jet velocity. It is observed that the choice of the reference value shifted the value of the flight velocity exponent at all inlet angles. It is observed that at lower inlet angles, the exponent values were close to those obtained for single stream jets. But at higher inlet angles, the exponent value was not as high as was obtained in single stream jet analysis. Rather, the exponent values remained nearly constant at higher inlet angles. The heated core jet predicts even lower values of the exponent as less reduction in the OASPL was observed in this case. The values again remained low at higher inlet angles. This study has been performed with $M_{cf} = 0.0$ and 0.2 and thus needs more data for a more reliable evaluation of the flight velocity expo-
The scaling of the spectra provided a good collapse of the spectra at all frequencies. It should be noted that the choice of the reference velocity does not affect the spectra scaling.

• The effect of pylon on the flow characteristics of the dual-stream nozzle has been studied by including a simplified pylon structure with the baseline dual-stream nozzle. The mean averaged velocity and instantaneous vorticity contours showed the effect of the pylon on the jet plume. The flow separated at the pylon end and thus created higher velocity gradients in that region. The axisymmetric nature of the nozzle was lost which is visible by comparing the flow characteristics in pylon and out of pylon planes. This will result in the azimuthal variation of the noise spectra for a particular radial location and polar angle.

6.2 Suggestions for Future work

This thesis presents the jet noise simulation results for single and dual-stream jets using a hybrid turbulent flow simulation method for free-shear layer flows. The flow and noise simulations show encouraging results which can be used for further studies such as to determine noise source locations and test various jet noise reduction mechanisms. There are several areas where further improvements and further study can be performed related to high-subsonic jet noise.

The grid size in the present work was restricted due to limitations in the available computer resources. Better grid resolution will be able to capture the high frequency content in the noise spectra. Using more CPUs will reduce the computation time and longer time samples can be recorded to obtain smoother noise spectra. The use of additional convergence acceleration methods such as the multi-grid approach or an implicit time integration method would also reduce the computation time. Therefore, one of the major issues in unsteady jet noise simulations is the development of a highly efficient parallel solver and the availability of computing resources.
There have been several suggestions regarding the shape and size of the acoustic data surface and this issue needs to be investigated further. Some of the proposed modifications in the surface include: (1) an open downstream end of the surface [30], (2) averaging data on several downstream end caps placed parallel to each other at a short separation distance [22, 99], (3) use of a tapered design where the surface tightly follows the jet plume spreading [100].

The mechanisms involved in the initial development of the jet flow need to be investigated further, since this affects the potential core length of the jet as well as the radiated noise. The present study has under-predicted the potential core lengths, and the reason of this needs to be investigated. More data should be collected for different co-flow Mach numbers and jet exit conditions to form a database to further assess the flight effects on jet noise. In particular, the present study includes only two co-flow Mach numbers for dual-stream jets which does not provide a very reliable estimate of the flight effects. Simulations should be run at other Mach numbers to obtain a better fit.

A simplified version of the pylon structure is used in the current research. Future research should include a more realistic pylon geometry to study the effects on noise with realistic configurations. The effect of other installation devices such as flaps on the jet noise can also be investigated. These cases will have more complex grids and will require more CPU time.
Appendix A

Governing Equations in General Curvilinear Coordinates

This sections lists the governing equations used to solve the flow-field. The Navier-Stokes equations, named after Claude-Louis Navier and George Gabriel Stokes, and the eddy viscosity equation (See Chapter 2 Section 2.1) are written in generalized coordinates to solve the equations on the multi-block structured grid. The grid is transformed using a grid transformation matrix from physical domain \((x, y, z)\) to computational domain \((\xi, \eta, \zeta)\) and the governing equations are re-written to comply with the computational domain coordinates.

The governing equations are presented here in three dimensions. The equations for two dimensional or one dimensional calculations can be derived by setting the derivatives in \(\zeta\), or \(\eta\) and \(\zeta\) directions to zero respectively. The generalized coordinates \((\xi, \eta, \zeta)\) are a function of physical coordinates \((x, y, z)\) and can be written as equation [A.1]

\[
\begin{align*}
\xi &= \xi(x, y, z) \\
\eta &= \eta(x, y, z) \\
\zeta &= \zeta(x, y, z)
\end{align*}
\]  
(A.1)
and the Jacobian transformation matrix used to relate the computational and physical coordinates is defined as given in equation \( A.2 \):

\[
J = \det \frac{\partial(x, y, z)}{\partial(\xi, \eta, \zeta)} = \begin{vmatrix}
\xi_x & \xi_y & \xi_z \\
\eta_x & \eta_y & \eta_z \\
\zeta_x & \zeta_y & \zeta_z \\
\end{vmatrix}^{-1}
\]

(A.2)

where \( \partial \) denotes the partial derivative, \( \det \) denotes the determinant and subscripts represent partial derivatives. The unit normal vector at a grid point along a constant \( \xi \) value, denoted by \( \vec{l}_\xi \), is defined as:

\[
\vec{l}_\xi = \left\{ \frac{\xi_x}{|\nabla\xi|}, \frac{\xi_y}{|\nabla\xi|}, \frac{\xi_z}{|\nabla\xi|} \right\}
\]

(A.3)

where, \(|\nabla\xi| = \sqrt{\xi_x^2 + \xi_y^2 + \xi_z^2}\). Similar forms can be used to derive unit normal vectors in \( \eta \) and \( \zeta \) directions. These values are typically used in boundary conditions implementation, for example, to calculate the flow direction (inflow or outflow) on a computational face.

The transformation from physical to computational domain is applied to the dimensional unsteady RANS equations (equation 2.4) and the transformed equations are obtained as shown in equation \( A.4 \):

\[
\frac{\partial \hat{Q}}{\partial t} + \frac{\partial \hat{E}}{\partial \xi} + \frac{\partial \hat{F}}{\partial \eta} + \frac{\partial \hat{G}}{\partial \zeta} = \frac{\partial \hat{E}_V}{\partial \xi} + \frac{\partial \hat{F}_V}{\partial \eta} + \frac{\partial \hat{G}_V}{\partial \zeta}
\]

(A.4)

The inviscid and viscous terms are separated in equation \( A.4 \) for convenience. Variables with hat, for example \( \hat{Q} \), are in generalized curvilinear coordinates. They are a function of flow variables in Cartesian coordinates and are related to them.
as shown in equations A.5 - A.11

\[ \hat{Q} = \frac{Q}{J} = \frac{1}{J} \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ \rho e \end{pmatrix} \]  (A.5)

\[ \hat{E} = \frac{\xi_x E + \xi_y F + \xi_z G}{J} = \frac{1}{J} \begin{pmatrix} \rho U \\ \rho U u + \xi_x p \\ \rho U v + \xi_x p \\ \rho U w + \xi_x p \\ (\rho e + p)U \end{pmatrix} \]  (A.6)

\[ \hat{E} = \frac{\eta_x E + \eta_y F + \eta_z G}{J} = \frac{1}{J} \begin{pmatrix} \rho V \\ \rho V u + \eta_x p \\ \rho V v + \eta_x p \\ \rho V w + \eta_x p \\ (\rho e + p)V \end{pmatrix} \]  (A.7)

\[ \hat{G} = \frac{\zeta_x E + \zeta_y F + \zeta_z G}{J} = \frac{1}{J} \begin{pmatrix} \rho W \\ \rho W u + \zeta_x p \\ \rho W v + \zeta_x p \\ \rho W w + \zeta_x p \\ (\rho e + p)W \end{pmatrix} \]  (A.8)

\[ \hat{E}_V = \frac{\xi_x E_V + \xi_y F_V + \xi_z G_V}{J} = \frac{1}{J} \begin{pmatrix} 0 \\ \xi_x A + \xi_y B + \xi_z C \\ \xi_x \tau_{xx} + \xi_y \tau_{xy} + \xi_z \tau_{xz} \\ \xi_x \tau_{xy} + \xi_y \tau_{yy} + \xi_z \tau_{yz} \\ \xi_x \tau_{xz} + \xi_y \tau_{yz} + \xi_z \tau_{zz} \end{pmatrix} \]  (A.9)
In the equations A.6 - A.11, $U$, $V$ and $W$ are the contravariant velocities in the three generalized coordinate directions. The variables $\rho$, $p$ and $e$ are the density, pressure and total energy respectively. $u$, $v$ and $w$ are the velocity components in the three directions. The contravariant velocities can be calculated as:

\[
U = \xi_x u + \xi_y v + \xi_z w \quad (A.12)
\]
\[
V = \eta_x u + \eta_y v + \eta_z w \quad (A.13)
\]
\[
W = \zeta_x u + \zeta_y v + \zeta_z w \quad (A.14)
\]

The terms $A$, $B$ and $C$ in equations A.9 - A.11 are defined as:

\[
A = u \tau_{xx} + v \tau_{xy} + w \tau_{xz} \quad (A.15)
\]
\[
B = u \tau_{xy} + v \tau_{yy} + w \tau_{yz} \quad (A.16)
\]
\[
C = u \tau_{xz} + v \tau_{yz} + w \tau_{zz} \quad (A.17)
\]

The viscous stress and heat transfer terms in the equations A.9 - A.11 and A.15 - A.17 can be written in dimensional form as:

\[
\tau_{xx} = \frac{2}{3} \mu \left( \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{\partial w}{\partial z} \right) \quad (A.18)
\]
\[
\tau_{yy} = \frac{2}{3} \mu \left( -\frac{\partial u}{\partial x} + 2 \frac{\partial v}{\partial y} - \frac{\partial w}{\partial z} \right) \quad (A.19)
\]
\[ \tau_{zz} = \frac{2}{3} \mu \left( -\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} + 2 \frac{\partial w}{\partial z} \right) \]  
(A.20)

\[ \tau_{xy} = \tau_{yx} = \mu \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \]  
(A.21)

\[ \tau_{yz} = \tau_{zy} = \mu \left( \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right) \]  
(A.22)

\[ \tau_{xz} = \tau_{zx} = \mu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \]  
(A.23)

\[ q_x = -\frac{C_P \mu}{Pr} \frac{\partial T}{\partial x} \]  
(A.24)

\[ q_y = -\frac{C_P \mu}{Pr} \frac{\partial T}{\partial y} \]  
(A.25)

\[ q_z = -\frac{C_P \mu}{Pr} \frac{\partial T}{\partial z} \]  
(A.26)

where \( C_P \) is the specific heat at constant pressure, \( Pr \) is the Prandtl number and \( \mu \) is the coefficient of dynamic viscosity. The derivatives such as \( \partial / \partial x \) can be calculated from the generalized coordinates as:

\[ \frac{\partial}{\partial x} = \xi_x \frac{\partial}{\partial \xi} + \eta_x \frac{\partial}{\partial \xi} + \zeta_x \frac{\partial}{\partial \xi} \]  
(A.27)

\[ \frac{\partial}{\partial y} = \xi_y \frac{\partial}{\partial \xi} + \eta_y \frac{\partial}{\partial \xi} + \zeta_y \frac{\partial}{\partial \xi} \]  
(A.28)

\[ \frac{\partial}{\partial z} = \xi_z \frac{\partial}{\partial \xi} + \eta_z \frac{\partial}{\partial \xi} + \zeta_z \frac{\partial}{\partial \xi} \]  
(A.29)

The partial derivatives can be calculated by using spatial discretization schemes such as seven point DRP scheme or second order central difference scheme. The present work uses DRP scheme as described in Chapter 2 Section 2.2. The grid transformation matrices are calculated using the free-stream preservation method.
Appendix B

Boundary Conditions

The boundary conditions used at the boundaries of the computational domain and at the block interfaces for physical accuracy of the solution are described in this Appendix. The details of the following boundary conditions are presented next: Riemann Invariant Boundary Conditions, Radiation Boundary Conditions, Methods of Characteristics Boundary Conditions, and Total Boundary Conditions.

B.1 Riemann Invariant Boundary Conditions

Riemann invariants, named after Bernhard Riemann, are mathematical transformations made on a system of quasi-linear first order partial differential equations to make them more easily solvable. Riemann boundary condition formulation is based on this property of the Riemann invariants. The characteristic equations of the one-dimensional Euler equations show that the entropy, $s$ and the Riemann invariants are constant along the characteristics in $x - t$ plane traveling with speeds $u - c$ and $u + c$. $c$ is the adiabatic speed of sound calculated as $c^2 = \gamma p/\rho$. These properties of the waves exiting or entering the boundaries can be used to determine the flow direction at the boundary and the flow variables can be calculated accordingly. Equations [B.1] show the one-dimensional Euler equations which will be used to derive the Riemann invariants. Similar analysis can be performed for
the three-dimensional equations.

\[
\begin{align*}
\frac{\partial}{\partial t} \begin{pmatrix} q_1 \\ q_2 \\ q_3 \end{pmatrix} + \begin{pmatrix} 0 & 1 & 0 \\ \frac{-3}{2} \rho u^2 & (3 - \gamma) u & (\gamma - 1) u \\ -\{\gamma e_{\text{tot}}u + (\gamma - 1)u^3\} & \{\gamma e_{\text{tot}} + \frac{3}{2}(1 - \gamma)u^2\} & \gamma u \end{pmatrix} \begin{pmatrix} q_1 \\ q_2 \\ q_3 \end{pmatrix} = 0
\end{align*}
\]

(B.1)

where \( q_1 = \rho \), \( q_2 = \rho u \), \( q_3 = \rho e_{\text{tot}} \), \( e_{\text{tot}} \) is the total energy. The eigenvalues of this system of equations are listed in equation (B.2)

\[
\begin{align*}
\lambda_1 &= u - c \\
\lambda_2 &= u \\
\lambda_3 &= u + c
\end{align*}
\]

(B.2)

with the corresponding eigenvectors shown in equation (B.3)

\[
\begin{align*}
e_1 &= \begin{pmatrix} 1 \\ u - c \\ h_{\text{tot}} - cu \end{pmatrix} \\
e_2 &= \begin{pmatrix} 1 \\ u \\ \frac{1}{2}u^2 \end{pmatrix} \\
e_3 &= \begin{pmatrix} 1 \\ u + c \\ h_{\text{tot}} + cu \end{pmatrix}
\end{align*}
\]

(B.3)

where total specific enthalpy is \( h_{\text{tot}} = e_{\text{tot}} + p/\rho \). The eigenvectors depend on the state \( q = (q_1, q_2, q_3) \) and are listed here for the Euler equations.

Typically one can express integral curves not only as integrals along the eigenvectors of the Jacobian, but also curves for which some special scalars are constant. In the three dimensional parameter space of the \( q = (q_1, q_2, q_3) \) state vector, each curve is defined by two of such scalars. Such scalar fields are called Riemann invariants of the characteristic family. One can regard these integral curves now as the crossing lines between the two contour curves of the two Riemann invariants. The value of each of the two Riemann invariants now identifies each of the characteristic integral curves. For the eigenvectors given in equation (B.3) the Riemann invariants are given as:

\[
s, u + \frac{2c}{\gamma - 1}
\]

(B.4)
Equations [B.4] and [B.6] represent acoustic waves (excluding shock waves). Equation [B.5] represents the entropy wave which means that entropy can change along this wave. So while the density may increase along this integral curve, the $e_{tot}$ will then decrease enough to keep the pressure constant. This means that the entropy will also go down, hence the term *entropy wave*.

Another wave, across which there is a jump in flow parameters, is a shock wave which can be either from the one-characteristic family or from the three-characteristic family. However, shock waves are waves for which the Riemann invariants are no longer perfectly invariant. In particular the entropy will no longer be constant over a shock front. Nevertheless, shock fronts can still be associated to either the one-characteristic or three-characteristic family. The states on both sides of the shock front however, do not lie on the same integral curve. They lie instead on a *Hugoniot locus*.

In implementing these boundary conditions, the flow variables are reset on the boundary using the constant behavior of the Riemann invariants on the characteristic waves. The velocity component normal to the boundary surface is used in the one-dimensional analysis. For example, the unit normal vector at the boundary with constant $\xi$ can be calculated as:

$$\hat{n}_\xi = \frac{\xi_x \hat{i} + \xi_y \hat{j} + \xi_z \hat{k}}{\sqrt{\xi_x^2 + \xi_y^2 + \xi_z^2}}$$  \hspace{1cm} (B.7)

where $\xi_x$ represents derivative of $\xi$ with respect to $x$ etc. The unit normal points towards increasing $\xi$ i.e., into the computational domain at $\xi_{min}$ and out of the computational domain at $\xi_{max}$ boundary. The invariants in equation [B.4] and [B.6] are calculated at the boundary with $u = U_e$ or $U_\infty$ for outgoing and incoming waves respectively. The subscript $e$ denotes the extrapolated variables from inside the boundary using second order extrapolation. The primitive variables are used to obtain the information at the boundary. The incoming and outgoing Riemann
invariants equations can then be solved simultaneously to obtain \( u_n \) and \( c \). The sign of the normal component of the velocity \( u_n \) determines whether there is inflow or outflow at the boundary. The corresponding values of the entropy \( s \) should be used at the boundary. It is also assumed that the tangential component of the velocity propagates with \( u_n \). Now, all primitive variables can be constructed on the boundary as:

\[
\rho = \left( \frac{sc^2}{\gamma} \right)^{1/\gamma} \tag{B.8}
\]

\[
p = \frac{\rho c^2}{\gamma} \tag{B.9}
\]

The velocity components can be calculated using equations \([B.10]\) for incoming wave (inflow) and using equations \([B.11]\) for outgoing waves (outflow).

\[
u \equiv u_\infty - \left( \vec{U}_\infty \cdot \hat{n} - u_n \right) \hat{n}_\xi \tag{B.10}
\]

\[
v \equiv v_\infty - \left( \vec{U}_\infty \cdot \hat{n} - u_n \right) \hat{n}_\eta
\]

\[
w \equiv w_\infty - \left( \vec{U}_\infty \cdot \hat{n} - u_n \right) \hat{n}_\zeta
\]

\[
u \equiv u_e - \left( \vec{U}_e \cdot \hat{n} - u_n \right) \hat{n}_\xi \tag{B.11}
\]

\[
v \equiv v_e - \left( \vec{U}_e \cdot \hat{n} - u_n \right) \hat{n}_\eta
\]

\[
w \equiv w_e - \left( \vec{U}_e \cdot \hat{n} - u_n \right) \hat{n}_\zeta
\]

where \( \vec{U}_\infty \) is the free-stream velocity value and \( \vec{U}_e \) denotes the extrapolated value at the boundary.

### B.2 Radiation Boundary Conditions

A set of radiation and outflow boundary conditions were developed by Tam and Webb\textsuperscript{[62]} for the aeroacoustic problems of interest with uniform background flow.
Tam and Dong\cite{101} and Dong\cite{78} presented the set of radiation boundary conditions which relaxes the assumption of presence of mean flow. It is assumed that the boundaries are quite far from the acoustic sources and the only waves exiting the boundary are the acoustic waves. These equations are obtained by setting the entropy and vorticity waves to zero at the boundary.

Equations \ref{B.12} need to be satisfied at the boundaries for implementing the radiation boundary conditions. These equations are formulated for two-dimensional governing equations.

\begin{align*}
\frac{1}{a_\infty} \frac{\partial p}{\partial t} + \frac{\partial p}{\partial r} + \frac{2}{r} (p-p_\infty) &= 0 \\
\frac{1}{a_\infty} \frac{\partial u}{\partial t} + \frac{\partial u}{\partial r} + \frac{1}{r} u &= 0 \\
\frac{1}{a_\infty} \frac{\partial v}{\partial t} + \frac{\partial v}{\partial r} + \frac{1}{r} v &= 0 \\
\frac{1}{a_\infty} \frac{\partial \rho}{\partial t} + \frac{\partial \rho}{\partial r} + \frac{2}{r} (\rho-\rho_\infty) &= 0
\end{align*}

where $a_\infty$ is the ambient speed of sound, $p_\infty$ and $\rho_\infty$ are the ambient values of pressure and density. The similar formulation for three-dimensional waves is given in equations \ref{B.13}

\begin{align*}
\frac{1}{a_\infty} \frac{\partial p}{\partial t} + \frac{\partial p}{\partial r} + \frac{4}{r} (p-p_\infty) &= 0 \\
\frac{1}{a_\infty} \frac{\partial u}{\partial t} + \frac{\partial u}{\partial r} + \frac{2}{r} u &= 0 \\
\frac{1}{a_\infty} \frac{\partial v}{\partial t} + \frac{\partial v}{\partial r} + \frac{2}{r} v &= 0 \\
\frac{1}{a_\infty} \frac{\partial w}{\partial t} + \frac{\partial w}{\partial r} + \frac{2}{r} w &= 0 \\
\frac{1}{a_\infty} \frac{\partial \rho}{\partial t} + \frac{\partial \rho}{\partial r} + \frac{4}{r} (\rho-\rho_\infty) &= 0
\end{align*}

The outflow boundary conditions given in Tam and Webb\cite{62} for the linearized
two-dimensional Euler equation with a mean flow are listed in equations \[ \text{B.14} \]

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + u_o \frac{\partial \rho}{\partial x} &= \frac{1}{a_o^2} \left( \frac{\partial p}{\partial t} + u_o \frac{\partial p}{\partial x} \right) \\
\frac{\partial u}{\partial t} + u_o \frac{\partial u}{\partial x} &= - \frac{1}{\rho_o} \frac{\partial p}{\partial x} \\
\frac{\partial v}{\partial t} + u_o \frac{\partial v}{\partial x} &= - \frac{1}{\rho_o} \frac{\partial p}{\partial y} \\
\frac{1}{V(\theta)} \frac{\partial p}{\partial t} + \cos \theta \frac{\partial p}{\partial x} + \sin \theta \frac{\partial p}{\partial y} + \frac{p}{2r} &= 0
\end{align*}
\]

where the mean flow has density equal to \( \rho_o \), pressure \( p_o \) and velocity \( u_o \) in the x-direction. The pressure equation is written in polar coordinate system where \( r = \sqrt{x^2 + y^2} \) and \( \theta = \tan^{-1}(y/x) \).

### B.3 Methods of Characteristics Boundary Conditions

The boundary conditions used at the grid block interfaces are based on the methods of characteristics formation given by Thompson\[64\]. These characteristics are first derived for the Navier-Stokes equations and their specific applications at different types of block interfaces are then discussed.

The flux vector form of the governing compressible Navier-Stokes equations, transformed to the computational domain, can be expressed in generalized coordinates as:

\[
\frac{\partial \hat{Q}}{\partial t} + \frac{\partial \hat{E}}{\partial \xi} + C = 0
\]

For simplicity, derivatives in only \( \xi \) - direction are included in this analysis. The characteristic equations for the boundary conditions in \( \xi \) - direction will be derived here using equation \[ \text{B.15} \]. Similar analysis can be performed in \( \eta \) - and \( \zeta \) - directions. \( C \) is the source term that consists of viscous flux derivatives and the hat indicates the transformed properties in generalized coordinates. The generalized
\( \hat{Q} \) and \( \hat{E} \) vectors are given as:

\[
\hat{Q} = \frac{Q}{J} \quad (\text{B.16})
\]
\[
\hat{E} = \frac{1}{J}(\xi_x E + \xi_y F + \xi_z G)
\]

Using the chain rule, the temporal and spatial derivatives in equation B.15 can be expressed in terms of conservative flow variables:

\[
\frac{1}{J} \frac{\partial Q}{\partial t} + \left( \frac{\xi_x}{J} \frac{\partial E}{\partial Q} + \frac{\xi_y}{J} \frac{\partial F}{\partial Q} + \frac{\xi_z}{J} \frac{\partial G}{\partial Q} \right) \frac{\partial Q}{\partial \xi} + \left[ E \frac{\partial}{\partial \xi} \left( \frac{\xi_x}{J} \right) + F \frac{\partial}{\partial \xi} \left( \frac{\xi_y}{J} \right) + G \frac{\partial}{\partial \xi} \left( \frac{\xi_z}{J} \right) \right] + C = 0 \quad (\text{B.17})
\]

The vectors of the conservative variables and the Euler fluxes in Cartesian coordinates are given by equations B.18 - B.22:

\[
Q = \begin{pmatrix}
\rho \\
\rho u \\
\rho v \\
\rho w \\
\rho e
\end{pmatrix} \quad (\text{B.18})
\]

\[
E = \begin{pmatrix}
\rho u \\
\rho u^2 + p \\
\rho vu \\
\rho wu \\
(\rho e_t + p)u
\end{pmatrix} \quad (\text{B.19})
\]

\[
F = \begin{pmatrix}
\rho v \\
\rho uv \\
\rho v^2 + p \\
\rho wv \\
(\rho e_t + p)v
\end{pmatrix} \quad (\text{B.20})
\]
\[ G = \begin{bmatrix} 
\rho w \\
\rho uw \\
\rho vw \\
\rho w^2 + p \\
(\rho e_t + p)w 
\end{bmatrix} \quad (B.21) \]

\[ e_t = \frac{p}{(\gamma - 1)\rho} + \frac{u^2 + v^2 + w^2}{2} \quad (B.22) \]

\( J \) is the transformation Jacobian and \( e_t \) is the total energy per unit mass given in equation [B.22]. Equation [B.15] and [B.16] can now be rearranged in terms of conservative variables as:

\[ \frac{\partial Q}{\partial t} + A \frac{\partial Q}{\partial \xi} + \mathcal{C} = 0 \quad (B.23) \]

where,

\[ \mathcal{C} = \left[ E \frac{\partial}{\partial \xi} \left( \frac{\xi_x}{J} \right) + F \frac{\partial}{\partial \xi} \left( \frac{\xi_y}{J} \right) + G \frac{\partial}{\partial \xi} \left( \frac{\xi_z}{J} \right) \right] \quad (B.24) \]

The term \( \mathcal{C} \) does not contain derivatives of the flow variables in the \( \xi \) direction. Therefore, it can be evaluated directly. \( A \) can be calculated from equation [B.25] as:

\[ A = \xi_x \frac{\partial E}{\partial Q} + \xi_y \frac{\partial F}{\partial Q} + \xi_z \frac{\partial G}{\partial Q} \quad (B.25) \]

Matrix \( A \) is usually referred to as the “Jacobian of the inviscid flux”. Its eigenvalues have a clear physical meaning and are frequently referred to in numerical methods. The matrix \( A \) can be diagonalized as \( \Lambda = PAP^{-1} \). The matrix \( P^{-1} \) transforms the conservative variables into the characteristic variables and its inverse matrix \( P \) can diagonalize the flux-Jacobian matrices in the direction normal to the interface. The resulting diagonal terms of matrix \( \Lambda \) become the convection speeds of the characteristics. Equation [B.15] can now be written in terms of the diagonal matrix as:

\[ \frac{\partial Q}{\partial t} + P\Lambda P^{-1} \frac{\partial Q}{\partial \xi} + \mathcal{C} = 0 \quad (B.26) \]

Equation [B.26] can be rearranged to obtain a characteristic form in the direction normal to the interface where \( \xi \) keeps a constant value. The transformed characte-
teristic equation is a quasi-linear wave equation with a source term $S_c$.

$$\frac{\partial Q}{\partial t} = -P(L + S_c) \quad \text{(B.27)}$$

where $L = \Lambda P^{-1} \partial Q / \partial \xi$ and $S_c = P^{-1} \hat{C}$. The convection speeds which form the diagonal of matrix $\Lambda$ are given as:

$$\begin{pmatrix}
\bar{U} - c |\nabla \xi| & 0 & 0 & 0 & 0 \\
0 & \bar{U} & 0 & 0 & 0 \\
0 & 0 & \bar{U} & 0 & 0 \\
0 & 0 & 0 & \bar{U} & 0 \\
0 & 0 & 0 & 0 & \bar{U} + c |\nabla \xi|
\end{pmatrix} \quad \text{(B.28)}$$

where $\bar{U} = \xi_x u + \xi_y v + \xi_z w$ is the contravariant velocity, $\hat{U} = \xi_x u + \xi_y v + \xi_z w$ is the normalized contravariant velocity and $\xi_x = \frac{\xi_x}{|\nabla \xi|}$ gives the normal direction. The five diagonal components are labeled as $\lambda_{1,5} = \bar{U} \mp c |\nabla \xi|$ and $\lambda_{2,3,4} = \hat{U}$. They represent the propagation speeds of five characteristic waves. The spectral radius of matrix $A$, or the maximum speed, is $\lambda_{\xi} = |\bar{U}| + c |\nabla \xi|$. The formulation for matrices $P$, $P^{-1}$ and $S_c$ can also be found in Kim and Lee [63] and Du [70].

Equation B.27 can be interpreted as the propagation of five characteristic waves in a general curvilinear system. Different choices of the eigenmatrix $P$ will result in different forms of the characteristic waves. To avoid possible singularities in the governing equation, the eigenmatrix is defined as:

$$P = \begin{bmatrix}
\frac{\rho}{2c} & \tilde{\xi}_x & \tilde{\xi}_y & \tilde{\xi}_z & \frac{\rho}{2c} \\
\frac{\rho H}{2c} - \frac{\rho \bar{U}}{2} & \tilde{B}_{0x} & \tilde{B}_{0y} & \tilde{B}_{0z} & \frac{\rho H}{2c} + \frac{\rho \hat{U}}{2} \\
\frac{\rho u}{2c} - \frac{\rho \xi_x}{2} & u\tilde{\xi}_x & u\tilde{\xi}_y - \rho\tilde{\xi}_z & u\tilde{\xi}_z + \rho\tilde{\xi}_y & \frac{\rho u}{2c} + \frac{\rho \xi_x}{2} \\
\frac{\rho v}{2c} - \frac{\rho \xi_y}{2} & v\tilde{\xi}_x + \rho\tilde{\xi}_z & v\tilde{\xi}_y & v\tilde{\xi}_z - \rho\tilde{\xi}_x & \frac{\rho v}{2c} + \frac{\rho \xi_y}{2} \\
\frac{\rho w}{2c} - \frac{\rho \xi_z}{2} & w\tilde{\xi}_x - \rho\tilde{\xi}_y & w\tilde{\xi}_y + \rho\tilde{\xi}_x & w\tilde{\xi}_z & \frac{\rho w}{2c} + \frac{\rho \xi_z}{2}
\end{bmatrix} \quad \text{(B.29)}$$
and its inverse is given by equation \[ \text{(B.30)} \]

\[
P^{-1} = \begin{bmatrix}
\frac{|v|^2(\gamma-1)}{2pc} + \mathbf{\tilde{B}}_0^{i} & \frac{\gamma-1}{pc} & \frac{-u(\gamma-1)}{pc} & \frac{-v(\gamma-1)}{pc} & \frac{-w(\gamma-1)}{pc} \\
\frac{-\tilde{\xi}_x}{pc} & -\frac{\tilde{\xi}_y}{pc} & -\frac{\tilde{\xi}_z}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} \\
\frac{-\tilde{\xi}_y}{pc} & -\frac{\tilde{\xi}_z}{pc} & -\frac{\tilde{\xi}_x}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} \\
\frac{-\tilde{\xi}_z}{pc} & -\frac{\tilde{\xi}_x}{pc} & -\frac{\tilde{\xi}_y}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} & \frac{\gamma-1}{pc} \\
\end{bmatrix}
\]

\[ \mathbf{\tilde{B}}_0, \mathbf{\tilde{B}}_1 \text{ and } H \text{ are calculated from equations } \text{(B.31), (B.32) and (B.33) respectively.} \]

\[ \mathbf{\tilde{B}}_0 = |v|^2 \mathbf{\tilde{l}}_\xi + \rho \left( \mathbf{\tilde{v}} \times \mathbf{\tilde{l}}_\xi \right) \]  

\[ \mathbf{\tilde{B}}_1 = \left[ 1 - \frac{|v|^2(\gamma-1)}{2c^2} \right] \mathbf{\tilde{l}}_\xi - \frac{1}{\rho} \left( \mathbf{\tilde{v}} \times \mathbf{\tilde{l}}_\xi \right) \]  

\[ H = \frac{|v|^2}{2} \left( \frac{c^2}{\gamma-1} \right) \] \( (\text{Entropy}) \)  

where, the velocity magnitude \(|v| = \sqrt{u^2 + v^2 + w^2}\). \(\mathbf{\tilde{l}}_\xi\) is the unit normal vector at a grid point on the block interface, defined by \(\mathbf{\tilde{l}}_\xi = (\tilde{\xi}_x, \tilde{\xi}_y, \tilde{\xi}_z)\), where \(\tilde{\xi}_x = \xi_x / |\nabla \xi|, \tilde{\xi}_y = \xi_y / |\nabla \xi|, \tilde{\xi}_z = \xi_z / |\nabla \xi|\).

Equation \[ \text{(B.28)} \] represents the physical (entropy, vorticity and acoustic) waves with different convection speeds in the direction normal to the interface. An advantage of using the characteristic equation is that the incoming and the outgoing waves can be classified easily by the signs of their convection speeds. An incoming wave calculated within an isolated block is inaccurate because the information outside the block is not used for calculating the convection term in equation \[ \text{(B.28)}. \] Therefore, the incoming waves of one block should be compensated by the outgoing waves of the other adjacent block through the strict interface conditions.

A crucial step is matching the primitive variables on the left and the right side of the interface i.e., \((\rho^L, v^L, p^L) = (\rho^R, v^R, p^R)\). Superscripts \(L\) and \(R\) denote
left and right blocks respectively. This condition should be satisfied regardless of time. Also, the residue should match at the interface, i.e., \( \partial_t \left( \rho^L, v^L, p^L \right) = \partial_t \left( \rho^R, v^R, p^R \right) \). As a result, from equation B.27, it can be concluded that the time derivatives of the characteristic variables should be also matched:

\[
P^L (L + S_c)^L = P^R (L + S_c)^R
\]  

(B.34)

A conventional technique dealing with the characteristic equation is compensating the convection term. Either the left- or the right-hand convection term should be corrected by the other one. As already mentioned, the decision of which should be chosen is based on the sign of the convection speed, and the incoming convection term is to be corrected while the outgoing term is maintained. The positive sign represents the outgoing wave from the left block and the incoming wave to the right block. On the contrary, the negative sign represents the incoming wave to the left block and the outgoing wave from the right block. This kind of analysis is for realistic communication between the blocks.

\[
\begin{align*}
L^L_i &= L^R_i + S c^R_i - S c^L_i, \text{if } \Lambda^L_i = \Lambda^R_i < 0 \\
L^R_i &= L^L_i + S c^L_i - S c^R_i, \text{if } \Lambda^L_i = \Lambda^R_i > 0
\end{align*}
\]  

(B.35) (B.36)

Equation B.34 is the final form of the characteristic interface conditions. This correction is performed at every stage of the time-marching steps.

With the choice of the eigenmatrix \( P \) and its inverse \( P^{-1} \) in equations B.29 and B.30, the amplitudes of the characteristic waves in equation B.27 are given by equations B.37 and B.38.

\[
P^{-1} \delta Q =
\begin{bmatrix}
\frac{\delta p}{\rho c} - \delta \tilde{U} \\
(\delta \rho - \frac{\delta \tilde{U}}{\rho c}) \tilde{\xi}_x + (\delta \tilde{v} \times \tilde{l}_x)_x \\
(\delta \rho - \frac{\delta \tilde{U}}{\rho c}) \tilde{\xi}_y + (\delta \tilde{v} \times \tilde{l}_x)_y \\
(\delta \rho - \frac{\delta \tilde{U}}{\rho c}) \tilde{\xi}_z + (\delta \tilde{v} \times \tilde{l}_x)_z \\
\frac{\delta p}{\rho c} + \delta \tilde{U}
\end{bmatrix}
\]  

(B.37)
Now, the second, third and fourth components of \( L \) (they are actually the combination of entropy wave and vorticity wave in the physical domain) can be defined as a vector \( \vec{L} = (L_2, L_3, L_4) \) to simplify the governing equations:

\[
PL = A \frac{\partial Q}{\partial \xi} = \begin{bmatrix}
\lambda_1 \left[ \frac{1}{\rho c} \frac{\partial p}{\partial \xi} - \frac{\partial \vec{v}}{\partial \xi} \right]
\lambda_2 \left[ \left( \frac{\partial \rho}{\partial \xi} - \frac{1}{\rho c^2} \frac{\partial p}{\partial \xi} \right) \xi_x + \left( \frac{\partial \vec{v}}{\partial \xi} \times \vec{l} \right)_x \right]
\lambda_3 \left[ \left( \frac{\partial \rho}{\partial \xi} - \frac{1}{\rho c^2} \frac{\partial p}{\partial \xi} \right) \xi_y + \left( \frac{\partial \vec{v}}{\partial \xi} \times \vec{l} \right)_y \right]
\lambda_4 \left[ \left( \frac{\partial \rho}{\partial \xi} - \frac{1}{\rho c^2} \frac{\partial p}{\partial \xi} \right) \xi_z + \left( \frac{\partial \vec{v}}{\partial \xi} \times \vec{l} \right)_z \right]
\lambda_5 \left[ \frac{1}{\rho c} \frac{\partial p}{\partial \xi} + \frac{\partial \vec{v}}{\partial \xi} \right]
\end{bmatrix}
\]

(B.38)

For a given boundary condition, the same formulation as in Thompson\[64\] can be used to determine all components of \( L \) in equation B.38. Once found, they can be substituted into equations B.25 and B.39 to update the residuals at the boundaries. There is no singularity in this formulation.

This following sub-sections provides a way to simplify the governing equations of the block interface condition for very complex mesh topologies in general curvilinear coordinates. First, both blocks are assumed to have the same mesh orientation. Then, the equations are extended to situations with arbitrary mesh-orientation across the block interface. Finally, a high-order interpolation method is presented for the implementation of a non-matching block interface condition.
B.3.1 Fully-matching Block Interface Conditions

B.3.1.1 Same Mesh-Orientation

In the simplest case, both blocks are assumed to have the same orientation at the block interface, for example, the $J_{\text{max}}$-surface of the left block is connected to the $J_{\text{min}}$-surface of the right block, and so on. Starting with equation B.27, the residual form of the general N-S equations is recovered:

$$ \frac{\partial Q}{\partial t} = \text{Res} = -P(L + S_c) $$ (B.40)

This reveals the relationship between Res and $L + S_c$:

$$ \text{Res} = -P(L + S_c) $$ (B.41)

or

$$ L + S_c = -P^{-1}\text{Res} $$ (B.42)

Application of the characteristic interface condition will change $L$, and thus result in new residuals:

$$ \text{Res}^* = -P(L^* + S_c) $$ (B.43)

$$ = -P[(L^* + S_c) - (L + S_c) + (L + S_c)] $$

$$ = \text{Res} - P(L^* - L) $$

$$ = \text{Res} - P\Delta L $$

which shows that the block interface condition imposes corrections on the original residuals represented by $\Delta L$:

$$ \Delta\text{Res} = -P\Delta L $$ (B.44)

When the mesh-orientation remains the same across the block interface, two blocks
have the same normal vector at the interface, for instance \( \vec{L}_\xi = \vec{R}_\xi \), and the same eigen-matrix \( P \) and its inverse \( P^{-1} \). Therefore, the same block interface condition in equation \([B.34]\) is applied. The corrections to \( L \) for the left block can be written as:

\[
\Delta L^L = L^L - L^L
= (L^R + S^R - S^L) - L^L
= (L^R + S^R) - (S^L + L^L)
= -(P^{-1})^R \text{Res}^R + (P^{-1})^L \text{Res}^L
= -(P^{-1})^L (\text{Res}^R - \text{Res}^L)
\]

As a result, the block interface condition is developed in terms of the corrections to the residuals, which can be further written as a function of the difference of the residuals between the two blocks. Note that equation \([B.45]\) cannot be substituted into equation \([B.44]\) directly to update the residuals, because some terms of \( \Delta L^L \) might not be used when the characteristic waves are propagating from the left block to the right one.

A diagonal matrix \( M \) can be defined to describe the features of characteristic waves. For example, an identity matrix \( M = I \) means all the wave components are traveling supersonically from the right block to the left one. Therefore, all corrections to \( L \) go into the residual corrections for the left block. On the contrary, a zero matrix \( M = 0 \) means the reversed propagation of characteristic waves from the left block to the right one. Therefore, no correction appears for the left block.

With the use of the matrix \( M \), the block interface condition for the left block can be generalized as

\[
\text{Res}^{*L} = \text{Res}^L + (PMP^{-1})^L (\text{Res}^R - \text{Res}^L)
\]
B.3.1.2 Arbitrary Mesh-Orientation

Special treatment is required at the boundaries when the mesh orientations are not same at a block interface i.e., \( J_{\max} \)-surface of the left block does not match the \( J_{\min} \)-surface of the right block. In such a case, the possibilities are that the \( J_{\max} \)-surface of the left block might connect to the \( I_{\min} \) or \( K_{\min} \) surface or \( I_{\max} \), \( J_{\max} \) or \( K_{\max} \) surface of the right block. Since the normal vector \( \vec{l}_\xi \) (or \( \vec{l}_\eta \), \( \vec{l}_\zeta \)) always points to the direction in which the index of \( i \) (or \( j, k \)) grows, the unit normal vectors for both blocks are the same in the first two cases, but have opposite signs in the last three cases. Therefore, both blocks have a continuous mesh-orientation in the first two cases, but an opposite mesh-orientation in the last three possibilities.

By defining a transformation matrix \( T_{rl} \), the matrix \( P \) and its inverse \( P^{-1} \) in the right block can be transferred to the left block using equations B.47 and B.48:

\[
P^L = T_{rl}P^R \tag{B.47}
\]

\[
(P^{-1})^L = T_{rl}(P^{-1})^R \tag{B.48}
\]

where, the transformation matrix \( T_{rl} \) for two types of mesh-orientations, indicated by the superscripts \( s \) (same) and \( o \) (opposite) respectively, are given by equations B.49 and B.50:

\[
T_{rl}^c = \begin{bmatrix}
1 & \cdots & 0 \\
\vdots & 1 & \vdots \\
0 & \cdots & 1
\end{bmatrix} \tag{B.49}
\]

\[
T_{rl}^o = \begin{bmatrix}
0 & \cdots & 1 \\
-1 & \cdots & 1 \\
\vdots & -1 & \vdots \\
1 & \cdots & 0
\end{bmatrix} \tag{B.50}
\]
Therefore, the block interface condition becomes:

\[ L^L + P^L_c \equiv T_{rl}(L^R + P^R_c) \]  \hspace{1cm} (B.51)

In both cases, while the residual corrections \[B.43\] remains unchanged for the left block, the corrections to \( L \) should be re-calculated. According to equation \[B.51\],

\[
\Delta L^L = L^{\ast L} - L^L = [T_{rl}(L^R + P^R_c) - P^L_c] - L^L
\]
\[
= T_{rl}(L^R + P^R_c) - (P^L_c + L^L)
\]
\[
= -T_{rl}(P^{-1})^R\text{Res}^R + (P^{-1})^L\text{Res}^L
\]
\[
= -T_{rl}T_{rl}(P^{-1})^L\text{Res}^R + (P^{-1})^L\text{Res}^L
\]
\[
= -(P^{-1})^L\text{Res}^R + (P^{-1})^L\text{Res}^L
\]
\[
= -(P^{-1})^L(\text{Res}^R - \text{Res}^L)
\]

which is the same as equation \[B.45\]. Here, an interesting property of \( T_{rl} \) is used, i.e. \( T_{rl}T_{rl} = I \) for both cases. Consequently, the same formulations of \( \Delta L \) and \( \Delta \text{Res} \) are derived. The governing equation \[B.46\] can be used at the block interfaces in all the possible mesh orientation cases.

### B.3.2 Non-Matching Block Interface Conditions

At non-matching block interfaces, the grid points of one block may not coincide with grid points in the neighbor block. Therefore, before applying the block interface conditions discussed before, the flow variables need to be interpolated using the information from its neighbor block. It is important to use a high-resolution interpolation method to minimize the interpolation errors that might add up to the numerical errors of the discretization scheme. By decomposing the numerical error in wave number space, Tam[102] developed a high-accurate optimized interpolation scheme for acoustic problems. A quantitative error analysis showed that the optimized scheme is significantly superior to traditional methods when
the interpolation point is around the center of the stencil (which is excellent for overset grids). However, when the interpolation point is near the boundary of the stencil, which is the case in the current research, no significant improvement of this method is observed compared with the Lagrange interpolation. Therefore, the Lagrange interpolation method is used in the current work.

Figure B.1 shows the typical interpolation procedure in two-dimensional simulations for clarity. The similar method can be extended to three-dimensional simulations.

Because the grids in the physical domain might have a large aspect ratio and very large spatial gradients, the interpolation based on the Cartesian coordinates could introduce large errors. Therefore, the stencil is mapped into the computational domain before interpolation. A stencil with three grid points in each direction is used. The mapping is made such that an irregular 27-point stencil forms a $2 \times 2 \times 2$ cube in the computational domain, with the center at $(\xi, \eta, \zeta) = (0, 0, 0)$ as shown in figure B.1.

Using the products of Lagrange polynomials in the computational domain, the interpolation of any variables can be achieved using the equation B.53

$$f(\xi_0, \eta_0, \zeta_0) = \sum_{i=1}^{3} \sum_{j=1}^{3} \sum_{k=1}^{3} a_{ijk} f(\xi_i, \eta_j, \zeta_k)$$  \hspace{1cm} (B.53)
where, $f(\xi_i, \eta_j, \zeta_k)$ is the value of a function $f$ at the grid point $(\xi_i, \eta_j, \zeta_k)$, and the interpolation coefficient, $a_{ijk}$, is given by equation (B.54):

$$a_{ijk} = \prod_{l=1, l \neq i}^3 \frac{(\xi_0 - \xi_l)}{(\xi_i - \xi_l)} \prod_{m=1, m \neq j}^3 \frac{(\eta_0 - \eta_m)}{(\eta_j - \eta_m)} \prod_{n=1, n \neq k}^3 \frac{(\zeta_0 - \zeta_n)}{(\zeta_k - \zeta_n)} \quad (B.54)$$

Once the local coordinates $(\xi_0, \eta_0, \zeta_0)$ of the interpolation point inside the stencil are found, the values at the point can be evaluated according to equations (B.53) and (B.54). Particularly, the exact value will be recovered if the interpolation point happens to coincide with one point of the stencil. The coefficients depend only on the local grid coordinates, therefore they can be calculated and stored at the beginning of simulation.

The $(\xi, \eta, \zeta)$ coordinates of the interpolation point can be found by replacing the function value in equation (B.53) with the Cartesian coordinates and then solving the equation implicitly. To be specific, equation (B.53) can be rewritten as

$$f(\xi, \eta, \zeta) = \sum_{i=1, j=1, k=1}^{3,3,3} \tilde{a}_{ijk} \xi^{i-1} \eta^{j-1} \zeta^{k-1} \quad (B.55)$$

Therefore, replacing the function values by the $(x, y, z)$ coordinates of the interpolation results in

$$\begin{cases}
x(\xi, \eta, \zeta) = \sum_{i=1, j=1, k=1}^{3,3,3} \alpha_{ijk} \xi^{i-1} \eta^{j-1} \zeta^{k-1} \\
y(\xi, \eta, \zeta) = \sum_{i=1, j=1, k=1}^{3,3,3} \beta_{ijk} \xi^{i-1} \eta^{j-1} \zeta^{k-1} \\
z(\xi, \eta, \zeta) = \sum_{i=1, j=1, k=1}^{3,3,3} \gamma_{ijk} \xi^{i-1} \eta^{j-1} \zeta^{k-1}
\end{cases} \quad (B.56)$$

By substituting the Cartesian coordinates of the 27 points of the stencil into equation (B.56), the unknown coefficients $\alpha_{ijk}$, $\beta_{ijk}$ and $\gamma_{ijk}$ can be found, which gives the mapping of coordinates from the physical domain to the computational domain by

$$\begin{cases}
x = f_1(\xi, \eta, \zeta) \\
y = f_2(\xi, \eta, \zeta) \\
z = f_3(\xi, \eta, \zeta)
\end{cases} \quad (B.57)$$
Given the Cartesian coordinates \((x_0, y_0, z_0)\) of the interpolation point, the local coordinates \((\xi_0, \eta_0, \zeta_0)\) can be calculated using the Newton-Raphson method. Starting with an initial guess \((\xi_{01}, \eta_{01}, \zeta_{01})\), a better approximation \((\xi_{02}, \eta_{02}, \zeta_{02})\) can be estimated by

\[
\begin{bmatrix}
\xi_{02} - \xi_{01} \\
\eta_{02} - \eta_{01} \\
\zeta_{02} - \zeta_{01}
\end{bmatrix} = \left[ \frac{\partial(f_1, f_2, f_3)}{\partial(\xi, \eta, \zeta)} \right]^{-1} \begin{bmatrix} x_0 - x_{01} \\ y_0 - y_{01} \\ z_0 - z_{01} \end{bmatrix}
\]

This process is repeated until the required convergence is achieved. About 8 iterations have been found enough to reduce the error down to \(10^{-5}\). Therefore, 8 iterations have been used in the present research.

### B.4 Total Boundary Conditions

The total boundary conditions fix the total pressure and total temperature at the boundary and the static values are calculated using one-dimensional isentropic relations. These boundary conditions are used at the nozzle inlet to simulate the flow with desired jet exit Mach number and exit temperature. The total temperature and total pressure ratios are provided as in input condition for simulating the nozzle flow. Two different total conditions are implemented at the core and fan nozzle inlets of dual-stream jets. The isentropic relations are listed below:

\[
\frac{P}{P_t} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\frac{1}{\gamma - 1}} \quad (B.59)
\]

\[
\frac{T}{T_t} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{-1}
\]
Appendix C

Analytical Solutions

C.1 Periodic Pressure Pulse Propagation

The velocity potential at $\vec{x}$ generated by a point source at $\vec{x}_s$ in a moving stream with Mach number $M$ in the x-direction is given by,

$$\phi(x|x_s) = \frac{\gamma^2}{4\pi c^2 \bar{R}} \exp \left(ik \left(\bar{R} - M\gamma^2(x - x_s)\right)\right)$$  \hspace{1cm} (C.1)

where $k$ is the wavenumber, $\gamma = 1/\sqrt{1 - M^2}$ and $\bar{R}$ is given by equation (C.2).

$$\bar{R} = \gamma \sqrt{\gamma^2(x - x_s)^2 + (y - y_s)^2 + (z - z_s)^2}$$  \hspace{1cm} (C.2)

The pressure on the acoustic data surface in a moving stream is given by:

$$p' = i\rho_o \omega \phi - \rho_o \bar{u} \frac{\partial \phi}{\partial x}$$ \hspace{1cm} (C.3)

where $t$ is the time, $c$ is the speed of sound, $\omega = kc$ is the circular frequency, $\rho_o$ is the free stream density, $\bar{u}$ is the sum of the free-stream mean velocity and the acoustic particle velocity. The problem is now transformed into that of a moving source in a stationary stream. So the Mach number of the acoustic data surface is
equal to negative value of the free-stream Mach number and \( \bar{u} \) on the data surface is equal to the acoustic particle velocity alone. Set \( \bar{u} = v_s \) and substitute the values of velocity potential in equation \( C.3 \) to get:

\[
p' = \frac{\rho_o \gamma^2}{4\pi c^2 R^2} \left( v_s \bar{R}_x \cos \theta + \bar{R} \sin \theta \left| v_s k \bar{R}_x - v_s k \gamma^2 - \omega \right| \right)
\]  

(C.4)

where \( \bar{R}_x = \frac{\gamma^4 (x-x_s)}{R} \) and phase \( \theta = -\omega t + k \bar{R} - k \gamma^2 (x - x_s) \). \( v_s \) is the source velocity. Equation \( C.4 \) can be compared with the resultant noise obtained by performing the integration over the FW-H surface for a periodic source.

### C.2 Gaussian Pulse Propagation

The Gaussian distribution of a variable \( F \) can be obtained by equation \( C.5 \)

\[
F = \varepsilon F_o \exp \left[ -\frac{\ln(2)}{\alpha^2} \left( (x - x_o)^2 + (y - y_o)^2 + (z - z_o)^2 \right) \right]
\]  

(C.5)

where \( F \) can be \( p \) or \( \rho \), \( F_o \) is the free-stream value, \((x_o, y_o, z_o)\) is the location of the center of the pulse, \((x, y, z)\) are the coordinates inside the domain, \( \varepsilon \) is the amplitude of the pulse taken as \( x\% \) of the ambient value in case of density and \( x\%/\gamma \) in case of pressure, and \( \alpha \) is the half width of the pulse.

The two-dimensional analytical solution for the Gaussian pulse propagation in stationary frame of reference can be found in Tam and Webb [62]. It is reproduced here in equation \( C.6 \)

\[
p(x, y, t) = \frac{\varepsilon}{2\alpha_1} \int_0^\infty e^{-\omega^2/4\alpha^2} \times \cos (\omega t) J_0 (\omega \eta) \omega d\omega
\]  

(C.6)

where, \( \eta = \sqrt{(x - Mt)^2 + y^2} \), \( M \) is the uniform flow Mach number, equal to zero in case of stationary background flow, and \( J_0 \) is the Bessel function of order zero.
The analytical solution for the three-dimensional pulse is given by equation (C.7):

\[
p(x, y, z, t) = \frac{A}{2r} \left\{ (r - ct) e^{-\alpha_1 (r - ct)^2} + (r + ct) e^{-\alpha_1 (r + ct)^2} \right\}
\]  \hspace{1cm} (C.7)

where, \( A \) is the magnitude of initial perturbation, \( c \) is the speed of sound, \( \alpha_1 = \ln(2)/\alpha^2 \) and \( r \) is the location of the pulse.

### C.3 Gaussian Pulse Reflection from Solid Wall

The reflection of Gaussian pulse from a hard wall can be modeled using the analytical solution from the section [C.2] where both incident and reflected waves need to be considered. The combined solution can be written as shown in equations (C.8)-(C.9):

For \( x < ct \):

\[
p(t, \vec{x}) = \frac{1}{2} p_i(\vec{x} + ct) + \frac{1}{2} p_i(ct - \vec{x})
\]  \hspace{1cm} (C.8)

and for \( x > ct \):

\[
p(t, \vec{x}) = \frac{1}{2} p_i(\vec{x} + ct) + \frac{1}{2} p_i(ct - \vec{x})
\]  \hspace{1cm} (C.9)

where \( x = 0 \) is the wall and the wave travels in \( x \geq 0 \). The initial pressure distribution is given by \( p(0, x) \) and \( c \) is the ambient speed of sound.
Appendix D


D.1 Introduction of the Flow Solver

The EAGLEJet flow solver uses the Navier-Stokes equations in dimensional form on a multi-block structured grid in serial or parallel processing modes.

The code is written in Fortran 90/95 and C language and uses MPI (Message Passing Interface) for parallelization. The source code directory named “src” contains Fortran 90/95 and C files, one Makefile and one dependencies (Make.dependencies) file. The job is scheduled using Portable Batch System (PBS) scheduler. A script file (.sh), with the instructions for submitting the job, is written and an input file (input.nam) is used to read the directory paths of the particular test case.

The main features of the code are listed as:

- Compressible, 3D structured grid Navier-Stokes solver
- Written in the Fortran 90/95 language
- 4th order accurate, 5 stage LDDRK time integration (Low-Dissipation and Dispersion Runge-Kutta)
• Runge-Kutta $4^{th}$ order explicit time marching, steady time marching, local time stepping and dual-time stepping

• $4^{th}$ order accurate DRP finite differencing (Dispersion-Relation-Preserving), biased operator at the boundaries

• Explicit $6^{th}$ order low pass filtering

• Selective artificial dissipation based on pressure and density derivatives

• Buffer zone implementation

• Implicit Residual Smoothing

• Immersed Boundary Method

• Non-Matching mesh block interfaces

• Ffowcs Williams-Hawkings surface extraction integration and noise prediction

• Spalart-Allmaras one equation RANS turbulence model

• Detached Eddy Simulation (DES) implementation

• MPI (Message Passing Interface) parallel code

• Single and multi-block grid topology

• Input files in PLOT3D format - multi-block, unformatted and double precision format

• Output solution files are written in single-precision format to reduce the file size. Restart files are written in double-precision format to re-start the calculation from a previous run.

User inputs include, at a minimum, a PLOT3D grid file, a boundary conditions file (generated using the generic 3D solver in Gridgen® and edited to add the number of processors in each block and boundary conditions index) and an input.nam file which contains the input variables for the EAGLEJet solver. In addition to the
<table>
<thead>
<tr>
<th>BC No.</th>
<th>Boundary Condition Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>boundary between processors within a block (set within the code)*</td>
</tr>
<tr>
<td>-1</td>
<td>inter-block interface (set by Gridgen)*</td>
</tr>
<tr>
<td>-2</td>
<td>non-matching inter block boundary (set separately using a code)*</td>
</tr>
<tr>
<td>21</td>
<td>Inviscid adiabatic wall</td>
</tr>
<tr>
<td>23</td>
<td>Viscous adiabatic wall</td>
</tr>
<tr>
<td>24</td>
<td>No slip Isothermal wall</td>
</tr>
<tr>
<td>31</td>
<td>Total Inlet conditions for core nozzle</td>
</tr>
<tr>
<td>32</td>
<td>Total Inlet conditions for fan nozzle in dual-stream jets</td>
</tr>
<tr>
<td>41</td>
<td>Subsonic Inflow</td>
</tr>
<tr>
<td>42</td>
<td>Subsonic Outflow</td>
</tr>
<tr>
<td>43</td>
<td>Supersonic Inflow</td>
</tr>
<tr>
<td>44</td>
<td>Supersonic Outflow</td>
</tr>
<tr>
<td>52</td>
<td>Fixed pressure Method of Characteristics</td>
</tr>
<tr>
<td>53</td>
<td>Non-reflecting Method of Characteristics</td>
</tr>
<tr>
<td>55</td>
<td>Tam and Webb radiation boundary condition</td>
</tr>
<tr>
<td>62</td>
<td>Periodic boundary condition</td>
</tr>
<tr>
<td>63</td>
<td>Symmetry boundary condition (not fully implemented)</td>
</tr>
<tr>
<td>70</td>
<td>Riemann Boundary Condition for subsonic/supersonic inflow/outflow</td>
</tr>
<tr>
<td>71</td>
<td>Tam and Dong radiation boundary condition</td>
</tr>
<tr>
<td>81</td>
<td>2D Case (7 planes supplied in k direction)</td>
</tr>
<tr>
<td>82</td>
<td>1D Case (7 planes supplied in j and k direction)</td>
</tr>
<tr>
<td>90, 91, 92, 93</td>
<td>Different types of buffer zones</td>
</tr>
</tbody>
</table>

**Table D.1.** Boundary conditions implemented in the code (* not to be set by user)

PLOT3D grid file, EAGLEJet can read an “initial condition” file (Plot3D format) which contains the values of the flow variables in conservative form and starts the simulation with non-zero instantaneous field. If the initial conditions file is not provided, the code can initialize the flow field according to the instructions given in the input file.

A list of currently supported boundary conditions is given in Table D.1. Not all boundary conditions are fully programmed and implemented. If a boundary condition is not available, the user will be notified and the program will terminate after the warning.
D.2 Compiling and Running the Solver

D.2.1 Compiling the Code on Linux Clusters

The flowchart given in Figure D.1 shows the file and directory structure of the code. There are Makefiles in src/, mpi/ and serial/ directories. The Makefiles in the mpi/ and serial/ directories need to be executed to create the desired executable file which is placed inside the LESCode/ directory. The parallel version of the code makes LES.mpi executable and serial version of the code makes LES.serial executable file. The appropriate MPI module declaration should be made in the Makefile in the src/ directory before creating the executable. The input file ‘input.nam’ is placed in the same directory where the executable file is placed. The .nam files are simple text files that can be edited using any editor like vi, emacs etc. The empty spaces in these files do not matter.

The filenames and the directories are shown by different box colors in the flowchart. The Makefile and Make.dependencies files in the src/ directory point to the individual Fortran source files and their dependent files. First, make sure that the compiler you wish to use is in your path. To determine this, type

`:>which compilername`

where compiler name can be one of the standard Fortran compilers such as ifort, gfortran or f90. If no compiler is found, you can add the compiler to your path by modifying the .bashrc or .bash_profile file in your home directory to include the line:

`export PATH=/path/to/the/fortran/compiler:$PATH`

and then type

`:>source .bashrc`

to reference the new bash file.

The clusters used to compile and execute the code, at the time of the documentation of this user manual, are COCOA3, COCOA4 and Lionxo clusters at Penn
State University. All large calculations are performed on COCOA4 cluster at the department of Aerospace Engineering, Penn State University. It has in total 54 computation nodes and CentOS 5.2 Linux operating system. Each node has four dual-core Intel Xeon processors (CPU type: E7220, 2.93 GHz) and 16 GB RAMs. In COCOA4 cluster, the compiler path looks like this:

```
export PATH=/sbin:/usr/local/intel/fc/10.1.015/bin:/usr/local/mvapich_intel_10-0.1/bin:$PATH
```

It is a 32 bit Intel Fortran compiler version 10 for MPI (MVAPICH) which uses Infiniband network. To compile the code in serial mode, uncomment the line:

```
MPIModule = serialMPIModule.o
```

in the Makefile in the LESCode/src directory. Then go to the LESCode/obj/serial/
directory and type:

```bash
debug
directory and type:

```bash
directory and type:

```
make

and a serial executable file named LES.serial should be created in the LESCode/
directory. To run the serial code, type:

```bash
./LES.serial

To compile the code in parallel uncomment the line:

```bash
MPIModule = parallelMPIModule.o

in the Makefile in the LESCode/src directory and comment back the line:

```bash
MPIModule = serialMPIModule.o

Then go to the LESCode/obj/mpi/ directory and type:

```bash
make

and a parallel executable file named LES.mpi will be created in the LESCode/
directory and will be copied to any directory shown as ‘Case1’ in Figure D.1. If
you want to copy the executable to any other directory, add the following line in
the Makefile in the LESCode/obj/mpi/ directory:

```bash
$(CP) LES ../../Case1/LES.mpi or $(CP) LES ../../Case1/LES.serial

To submit a parallel job, you must create a PBS script file and have both the input
file and the executable file in the same directory.

A sample queue script file named QCocoa4.sh looks like:

```bash
#PBS -N testcase ! Submission name
#PBS -e errorfile ! Text file for any error output
#PBS -o outputfile ! Text file for other PBS output
#PBS -l nodes=1:ppn=2 ! Number of processors and number of nodes
#PBS -l walltime=120:00:00 ! Estimated run time (upper limit)
# This job’s working directory
```
echo Working directory is $PBS_O_WORKDIR
cd $PBS_O_WORKDIR

echo Running on host ‘hostname’
echo Time is ‘date’
echo Directory is ‘pwd’
echo This jobs runs on the following processors:
echo ‘cat $PBS_NODEFILE’

# Define number of processors
NPROCS='wc -l < $PBS_NODEFILE'
echo This job has allocated $NPROCS processors.

/usr/local/mvapich_intel_10-1.0.1/bin/mpirun PBS_JOBID=$PBS_JOBID -np $NPROCS -machinefile $PBS_NODEFILE ./LES.mpi >OUTPUT.txt

Then submit the job into the queue like this:

: > qsub QCocoa4.sh

Check the status of your running job by typing:

: > qstat

and you can see the progress of your running job by typing (code output is placed in OUTPUT.txt):

: > tail OUTPUT.txt

or

: > more OUTPUT.txt

or

: > less OUTPUT.txt

To submit the parallel executable file as a serial job in the background, simply make this change in the queue script:
Now, the code still uses MPI but with only one processor it effectively runs in serial mode. The minimum number of processors required to run a computation is equal to the number of the grid blocks. Therefore, the serial version can only be used with the cases with one grid block.

D.2.2 Preparing a Job to Run

The case file ‘input.nam’ specifies the case to be executed. This file must be at one of the two places:

- For a serial run using the ./LES.serial command, it must be in same directory as the LES.serial executable.
- For a parallel or serial run using a submission script, it must be in the same directory as the submission script.

The case files are located in the /validation/case_name. When the QCo-coa4.sh file is submitted, the executable located in LESCode/ runs. It reads the job file names from the ‘input.nam’ which points the code to the job directory: ..../validation/case_name/, namelist file: filename.nam, FW-H input file: PSFWH.in, and boundary conditions file: filenameBC.nam. filename.x (or .xyz or .grd) (found in the directory: ..../validation/ case_name/) is the PLOT3D grid file and filename.save is the initial conditions file (in PLOT3D solution file format). You can submit the script file from any directory, as long as the ‘input.nam’ file is in the same directory, the ‘input.nam’ contains the correct paths for the case files and the script points to the executable. The following needs to be specified in the ‘input.nam’ file:

```
&CaseIn
  CaseFolderName = ‘../validation/OneBlk/’ —>path to the case directory
  caseFileName = ‘oneblk.nam’ —>name of input file for the case
  caseBCFileName = ‘oneblkBC.nam’ —>name of boundary conditions file
  NozzleFileName = ‘nozzle_coord.dat’ —>file containing nozzle coordinates to spec-
```
ify initial conditions
FWHFileName = ‘PSFWH.in’ --> file containing information for acoustic field extraction
PointFile = ‘PNT.in’ --> file containing information for recording time history at points
/

The input paths highlighted in red are mandatory.

D.3 The Namelist File

This namelist file contains all the parameters needed to set up the case. All the parameters should be given in SI units. The parameters specified in the namelist file are next explained in detail:

D.3.1 &EnvironmentIn

- gridName - Name of the grid/mesh file
- debugLevel - controls how verbose the output of the code during execution is. Possible values are 1, 2, and 3 where 3 is the most verbose and 1 is the least. Default is 1.
- restartFlag - Turn this flag on to read a restart file and continue from where a previous run left off. The restart file must be in the job directory. This file is a PLOT3D Q file output by the last successfully completed run. The output file name after the last time step of the previous run is always located in the Results directory and named restart.q.
- restartName - Of the form “restart.q” or something similar. This is the name, in quotations, of the restart file in the case directory to read for the program restart.
• restartStep - An integer in terms of the time steps. It specifies the time step where the calculation is to be restarted.

• UNIFORMFLOWFLAG - This flag turns on the use of a uniform flow field for the mean flow. If this flag is on, the user must also specify the following:
  - PINF - the freestream pressure
  - RHOINF - the freestream density
  - MUINF - Dynamic viscosity of the fluid
  - MACH- the Mach number of the flow
  - Mach_sn- the Mach number at the fan jet exit
  - Mach_cf- the Mach number of the co-flow (forward flight effect)
  - $V_x, V_y$ and $V_z$ - the x, y, and z component of the flow direction

• viscflag - Whether or not to calculate the viscous terms during the integration. Default is .true.. This should be set to .false. for inviscid calculations.

• ArtDissFlag - If .true., selective artificial dissipation is used in the code.

• TamCoeffFlag - If .true., the dissipation coefficients developed by Tam are used. Currently it is set to .false. and should not be used.

• LowPassFilterFlag - applies the low pass filter routine of Visbal and Gaitonde. Can be used with NBLOWPASSFILTERS to define the number of times to apply the filter at each substep. Default is one time.

• DualTimeStepFlag - If this flag is set to true, the solver uses dual time stepping for time marching. This accelerates the rate of convergence. If this is true, user should also provide the desired physical time step (dt_P), desired value of residual at convergence (ep_residual) and the maximum number of sub-iterations (nSubIterMax).

• LocalTimeStepFlag - If this flag is true (for viscous cases), the solver will use local time step instead of global time step for time marching. Generally
used in viscous cases when there are large differences in grid spacing.

- TIMEMARCHSCHEME - 1 for LDDRK time marching, 2 for Euler time marching, 3 for Runge-Kutta 4 stage time marching, 4 for steady time marching, 5 for dual-time stepping. Default is 3.

- METRICSSCHEME - Specifies the method used to calculate grid metrics. It is equal to 1 for freestream preservation method and 2 for standard method.

- TurbSpalartFlag - Set to .true. if Spalart-Allmaras turbulence model needs to be turned on. Default value is .false.

- PeriodicSourceFlag - Set to .true. if a periodic source is desired at the center of the computational domain. Should provide &PSourceIn and &FWHSurfIn if this flag is true. Default value is .false.

- PeriodicGaussFlag - Set to .true. if a periodic Gaussian pulse is desired at the center of the computational domain. Presently the wavenumber of the pulse is set to 0.2, the half-width of the Gaussian pulse is set to 5.0, and the perturbation values are 0.05 times the ambient values (Later, these values will be included in the input . nam file to be entered as an user input). Default value of this flag is .false.

- idimension - Sets the dimension in which the distance needs to be calculated for DES implementation. The value of 2 corresponds to the j-direction and 3 correspond to the k-direction. Set it to 2 for quasi-2D problems and to 3 for 3D problems.

- inputFileFlag - Set to true if an initial condition file is provided. Set to false if flow is initialized within the code.

- iCase - An integer specifying the flow initialization within the code if inputFileFlag is false. Presently the iCase number can have the value between 1 to 10.

- P_inl_total - Total pressure specified at the nozzle inlet in jet flow cases

- T_inl_total - Total temperature specified at the nozzle inlet in jet flow cases
• $P_{inl\_total\_sn}$ - Total pressure specified in jet cases at secondary (fan) nozzle inlet

• $T_{inl\_total\_sn}$ - Total temperature specified in jet cases at secondary (fan) nozzle inlet

• $ibuf$ - 1 or 0 depending on whether the buffer zone is implemented in the boundary conditions file or not.

• $sample\_flag$ - True if either or both FWH sampling and specific point(s) time history is required.

• $issmooth$ - 0 or 1 depending on whether the Implicit Residual Smoothing is turned off or on.

• $IBMFlag$ - 1 or 0 depending on whether the computation uses Immersed Boundary Method or not.

• $SolutionPath$ - A character string locating the directory, relative to the case directory, where the solutions will be written to.

• $tMin$ - Start time for the time marching.

• $tMax$ - End time for the time marching.

• $delt$ - The time step value for the integration. This should not be used if CFL number is specified.

• $CFL$ - If specified, the largest time step allowable by stability criteria is calculated and used. Should not be used if specifying $delt$.

• $ntSteps$ - The number of time marching steps required in the current simulation. If restarting from a previous run, this is the number of additional timesteps to take beyond the integer restartStep. After the solution has marched the required number of timesteps, a final solution file will be written along with a restart file “restart.q”.

• $ntWrite$ - The solution files are written at an interval of these many time steps. If not specified, no intermediate solution files will be written.
<table>
<thead>
<tr>
<th>Initial Conditions No.</th>
<th>Flow Initialization Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2D Gaussian pulse with center at (0,0) and half width equal to 3.0</td>
</tr>
<tr>
<td>2</td>
<td>3D Gaussian pulse with center at (0,0,0)</td>
</tr>
<tr>
<td>3</td>
<td>Uniform flow case: flat plate or airfoil case</td>
</tr>
<tr>
<td>4</td>
<td>3D nozzle: nozzle blocks no. 10 to 14</td>
</tr>
<tr>
<td>5</td>
<td>3D nozzle: nozzle blocks no. 5 to 9 initialized with isentropic flow</td>
</tr>
<tr>
<td>6</td>
<td>Immersed Boundary Method (uniform flow)</td>
</tr>
<tr>
<td>7</td>
<td>3D nozzle: nozzle blocks no. 5 to 9</td>
</tr>
<tr>
<td>8</td>
<td>Dual-stream nozzle: core nozzle blocks 5 to 9, fan nozzle blocks 1 to 4</td>
</tr>
<tr>
<td>9</td>
<td>3D nozzle: nozzle blocks no. 5 to 9</td>
</tr>
<tr>
<td>10</td>
<td>3D nozzle: nozzle blocks no. 1 to 5</td>
</tr>
</tbody>
</table>

**Table D.2.** Initial conditions options implemented in the code.

- **ntRestartWrite** - The restart files are written after an interval of these many time steps. The restart files are required to start a calculation from an intermediate solution.

- **dtWrite** - This is the amount of time between writing the solution files. Should not be used with ntWrite.

- **nSubIterMax** - Number of sub-iterations in fictitious time used in dual time stepping mode.

- **dt_P** - Physical time step used in dual time stepping.

- **timestep_dpdt** - Time step at which the pressure gradient $dp/dt$ should be written in an output file.

- **nmean_0** - Number of time steps that should be used in calculating the time mean flow values.

The default output is a PLOT3D solution file consisting of the instantaneous flow variables in conservative form. The options currently implemented in the code to initialize the flow are listed in Table D.3.1.
D.3.2 &GridIn

Within the GridIn section, integers can be specified such as the number of boundary conditions, number of initial conditions, number of specific points to save the solution, and number of acoustic data surfaces. If any of these integers are specified, then immediately following the GridIn section must be their respective namelist, in the order listed above.

- GridNumber - this is the integer number of the grid in the GridPath file which the solution is performed on. The default is 1.
- nbIC - an integer specifying the number of initial conditions to be declared on the domain. An &ICIn section must immediately follow the &BCIn sections for each initial condition desired.

D.3.3 &MultiBlockParam

- NBLK - Number of blocks in the grid

D.3.4 &BlkInfo

The block information needs to be specified for each block present in the multi-block grid.

- BlkNum - Block number (goes to 1 to N where N is the total number of blocks)
- nProcsBlk - Number of processors to be used in each block

D.3.5 &bufferZone

- npts_buffer - Number of grid points up to which the buffer zone is implemented
• bufc1 - Constant value used in buffer zone
• bufc2 - Constant value used in buffer zone

D.3.6 &ArtDiss

• d_2 - damping coefficient 1
• d_6 - damping coefficient 2

D.3.7 &IRS

• RCFL - Modified CFL number to be used in Implicit Residual Smoothing. Usually a value of 2.5 is used. No need to change this value.

D.3.8 &IBM

• IMFlagFile - Name of the marker file used in Immersed Boundary Method. This file contains the value of a flag (either 1 or 0) for the whole computational grid depending on whether the grid point lies inside or outside the immersed solid body.

D.3.9 &ICIn

The initial conditions on the flow field must be stored as a double precision Fortran unformatted Q file. The solution file must be of the same dimension as the input grid. If inputFileFlag in EnvironmentIn namelist is set to false, then the initial flow field is calculated within the code.

• Filename - a character string specifying the name of the initial condition file, relative to the directory the case namelist is in. For example, ‘./initial.save’.
D.3.10  &PointIn

- isample - There are 8 combinations for isample. It has two integers: the first integer specifies the type of FWH sampling, and the second integer specifies the type of point sampling.

  * isample = 10, 11 or 12: FWH sampling not done on the grid surface
  * isample = 20, 21 or 22: FWH sampling done on the grid surface
  * isample = 11, 21 or 01: if the block number and (i,j,k) indices of the point where the solution is desired are specified.
  * isample = 12, 22 or 02: if the (x,y,z) location of the desired point is specified.

D.3.11  &PSourceIn

These inputs are related to the periodic point source parameters.

- kwave - wavenumber of the periodic source to be used
- infZoneFac = Factor of influence of the periodic source i.e., the effect of periodic source is felt in the sphere with radius equal to (infZoneFac)\(\lambda\) with center at the origin of the pulse (\(\lambda\) = wavelength of periodic source)

D.3.12  &FWHSurfIn

These input variables are related to the extraction of data on Ffowcs Williams and Hawkings surface.

- nTeclt - Number of time steps at which a Tecplot® format (.plt) file is written to check the FWH integration surface
- nsmpL0 - Number of sampling time steps for which data have been written on the FW-H surface (used for restarting the sampling)
D.3.13 &Turb

- iturb - specifies the turbulence model. Valid values are 1 - Spalart Allmaras turbulence model and 0 - no turbulence model

- ides - Turns on (1, 2) and off (0) the Detached Eddy Simulation (DES) modeling. Should be turned on only when iturb is set to 1.

- restartTurbName - Restart file name for turbulent flow variables. Specify this path with the Results directory name.

D.4 The Boundary Conditions File and Point Sampling File

This file contains the block information, boundary conditions information and number of processors for each block. This file can be obtained by doing slight modification to the boundary conditions file obtained from the Gridgen® using generic 3D solver. A sample file in the case of two blocks is shown below:

```
1 !— generic solver id !
2 !— Number of blocks !
!– grid points and number of processors in i, j, and k direction–!
101 76 7 1 1 1

A !— block name !
6 !— number of boundary conditions on the block !
!– Number of points in i, j, and k directions and boundary condition type
1 101 76 1 1 1 81
101 1 76 1 7 7 81
1 1 76 1 7 1 55
101 101 76 1 1 7 55
1 101 1 1 1 7 55
1 101 1 1 -1 -7 -1
101 76 1 1 1 7 55

B
```
The last column is the boundary conditions index that needs to be set by the user except the inter-block and non-matching interface communication indices. The comments added here are not part of the actual file.

Sample point file for extracting time history of flow variables at several points in the flow field:

Sample type 1: the x, y, z location where the time history is required

```plaintext
# input file containing individual points at which the time histories of flow variables are required to output
# Number of grid points:
# NPNT
3

# grid indices of the points:
# x, y, z
0.005  0.01  0.01
0.010  0.01  0.01
0.100  0.01  0.01
```

Sample type 2: The block number and the grid i, j, k indices where the time history is required

```plaintext
# input file containing individual points at which the time histories of flow variables are required to output
# Number of grid points:
# NPNT
30

# grid indices of the points:
# iblk, i, j, k
```
D.5 The Ffowcs Williams-Hawkings (FW-H) Surface Extraction File

The acoustic data surface extraction file is shown below for a three-dimensional one block case:

```plaintext
### input file for PSFWH code – this is a comment
!! nsmpl: total number of samples, nsio: sample after every these time steps,
nstart: start sampling after this timestep
300  2  50

#! start of grid information for FWH control surfaces
!! type I: surface type (1 sphere, 2 cylinder-open, 3 cylindrical-closed), Radius,
length, half_angle( in deg )
 1  10  20  0

!! Center of FWH surface
!! x0_s, y0_s, Z0_s, angle_x, angle_y, angle_z (in degrees)
0.0  0.0  0.0  0.0  0.0  0.0

!! Grid information on control surface.
!! Pnts in Theta, pnts in Phi or in polar and length direction, and pnts in radial
direction. must be 4*n+1
101  101  101

# type II: grid surfaces as the integration surfaces:
# NFWHSURFACE
6
# blkid imin imax jmin jmax kmin kmax
1  20  20  20  40  20  40
1  20  40  20  20  40  40
```
The FW-H surface can be extracted from several blocks by specifying the grid indices to be extracted from each block.

**D.6 Output Files**

The following solution files are written in the Results directory:

**sol.xyz** : Original grid file written in single precision floating point number to save disk space.

**1.q** : Initial flow conditions written in single precision for checking.

**sol.xxx.q** : Flow solution files written in single precision at xxx time steps.

**sol.eddy.xxx.q** : Variables associated with turbulent flow written in single precision at xxx time steps. These files are written only when the turbulence model flag is ON.

**sol.dqdt.xxx.q** : Gradients of flow variables written at xxx time steps for post-processing.

**restart.xxx.q** : Flow variables written in double precision at xxx time steps for restarting the calculations.

**restart.xxx.eddy.q** : Eddy values written in double precision to restart a turbulent flow calculation.

**restart.xxx.mean.q** : Mean flow variables written at xxx time steps in double precision and are used while restarting the calculation.

**restart.xxx.qm1.q** : Flow variables written at previous time step in double precision to be used in dual-time stepping method.
residual.tec : Maximum residual values written at every time step in Tecplot format.

time.tec : CPU time at every time step written in Tecplot format.

Point_x.dat : Time history for point x written in Tecplot format if required.

Point_all.dat : This file contains information about the grid points where time history is sampled.

sol_eddy.nam : File containing variable names for visualizing the solution in Tecplot.

dqdt.nam : File containing variable names for visualizing the solution in Tecplot.

Some or all of these output files are produced in a calculation.

D.7 PLOT3D File Formats

PLOT3D[103] is a computer graphics program designed to visualize the grids and solutions of computational fluid dynamics. PLOT3D file format is used throughout the present solver for CFD writing and reading grids and solution files. The format is multi-block, Fortran unformatted, double precision. The following is a segment of Fortran source code which generates the correct grid and solution Q files:

D.7.1 Grid File

integer, dimension(:,): nbi,nbj,nbk
integer::i,j,k
double precision, dimension(:,,:,3): grid
open(unit=unitNum,file=pathName,status='replace',form='unformatted')
write(outUnit) nblk
write(outUnit) (nbi(iblk),nbj(iblk),nbk(iblk),iblk = 0,nblk-1)
do iblk = 0, nblk-1
write(outUnit)
D.7.2 Solution Q File

double precision,dimension( ::, ::, :, 5) :: q
double precision :: mach, alpha, reyn, time
integer, dimension ( :: ) :: nbi, nbj, nbk
integer :: i, j, k, nblk, iblk, outUnit
open ( unit = outUnit, file = pathName, status = 'replace', form = 'unformatted' )
write ( outUnit ) nblk
write ( outUnit ) ( nbi ( iblk ), nbj ( iblk ), nbk ( iblk ), iblk = 0, nblk - 1 )

do iblk = 0, nblk - 1

write ( outUnit ) mach, alpha, reyn, time
write ( outUnit )
(( ( q ( i, j, k, 1 ) ), i = 1, nbi ( iblk ) ), j = 1, nbj ( iblk ) ), k = 1, nbk ( iblk ) ),
(( ( q ( i, j, k, 2 ) ), i = 1, nbi ( iblk ) ), j = 1, nbj ( iblk ) ), k = 1, nbk ( iblk ) ),
(( ( q ( i, j, k, 3 ) ), i = 1, nbi ( iblk ) ), j = 1, nbj ( iblk ) ), k = 1, nbk ( iblk ) ),
(( ( q ( i, j, k, 4 ) ), i = 1, nbi ( iblk ) ), j = 1, nbj ( iblk ) ), k = 1, nbk ( iblk ) ),
(( ( q ( i, j, k, 5 ) ), i = 1, nbi ( iblk ) ), j = 1, nbj ( iblk ) ), k = 1, nbk ( iblk ) )
end do
close ( outUnit )

D.7.3 Function File

double precision, dimension ( ::, ::, ::, :: ) :: func
integer, dimension ( :: ) :: nbi, nbj, nbk
integer:: outUnit, i, j, k, iblk, nfunc, nblk
open(unit=outUnit, file=pathName, status='replace', form='unformatted')
write(outUnit) nblk
write(outUnit) (nbi(iblk), nbj(iblk), nbk(iblk), nfunc, iblk = 0, nblk-1)
do iblk = 0, nblk-1
write(outUnit) (((func(i,j,k,1)), i=1, nbi(iblk)), j=1, nbj(iblk)), k=1, nbk(iblk)), &
(((func(i,j,k,2)), i=1, nbi(iblk)), j=1, nbj(iblk)), k=1, nbk(iblk)), &
(((func(i,j,k,3)), i=1, nbi(iblk)), j=1, nbj(iblk)), k=1, nbk(iblk)), &
(((func(i,j,k,4)), i=1, nbi(iblk)), j=1, nbj(iblk)), k=1, nbk(iblk)), &
(((func(i,j,k,5)), i=1, nbi(iblk)), j=1, nbj(iblk)), k=1, nbk(iblk)), &
-----
(((func(i,j,k,nfunc)), i=1, nbi(iblk)), j=1, nbj(iblk)), k=1, nbk(iblk))
end do
close(outUnit)
Bibliography


coustics Conference and Exhibit, 12-14 May, Hilton Head, South Carolina, 2003.


[82] “Subsonic Mach Number of 0.2 Past the Turbulent Flat Plate,” *NPARC Alliance Validation Archive (http://www.grc.nasa.gov/WWW/wind/valid/fpturb/fpturb.html)*.


