AN EXPERIMENTAL AND COMPUTATIONAL INVESTIGATION
OF AN ANNULAR REVERSE-FLOW COMBUSTOR

by

T. C. J. Hu, R. A. Cusworth and J. P. Sislian

November 1990

UTIAS Report No. 338
CN ISSN 0082-5255
AN EXPERIMENTAL AND COMPUTATIONAL INVESTIGATION
OF AN ANNULAR REVERSE-FLOW COMBUSTOR

by

T. C. J. Hu, R. A. Cusworth and J. P. Sislian

Submitted October 1990

©Institute for Aerospace Studies 1990

November 1990

UTIAS Report No. 338
CN ISSN 0082-5255
Abstract

The complex flowfield of a Pratt & Whitney Canada toroidal vortex annular reverse-flow sector-combustor has been investigated. The \( u \) and \( v \) components of the mean velocity and the corresponding turbulence intensities \( \overline{u'^2}, \overline{v'^2} \) and \( \overline{u'^v'} \) were mapped in detail for cold flow with and without fuel injection and hot flow conditions, with a two-component argon-ion laser Doppler velocimeter operated in dual-beam, forward scatter mode. The flowfield characteristics were identified from the LDV data and the flow visualization pictures substantiated the findings. Effects of heat addition on the flowfield were determined from comparisons of the cold and hot flows. Results show that combustion intensifies vortical and recirculating flows, increases the momentum of the fuel jet, locally laminarizes the fuel jet near the injector inlet, and increases the turbulence kinetic energy and turbulent stresses at the shear layers. The combustor aerodynamic developments in the flowfield with and without fuel injection are discussed. Predictions of the cold flow using a 2-D TEACH-type computer code demonstrated that the code can provide qualitative agreement for flow regions which are not strongly 3-D in nature, where a 3-D numerical model is required in resolving all the flow features realistically. Researchers and engineers will find the measured data essential in the understanding, evaluations and developments of combustor designs, and mathematical modeling of processes inside practical combustors.
Acknowledgement

The sector rig and combustor hardwares were provided by Pratt & Whitney Canada. The technical advice and support given by the manager of Hot-End Component Dr. P. Sampath, senior aerodynamicist Mr. A. Prociw of the Combustion Group, the chief of Testing Department Mr. A. Kong, and test engineer Mr. R. Acolacol are gratefully acknowledged.

The financial assistance received from the National Science & Engineering Research Council of Canada (NSERC) Strategic Grant G2016 is gratefully acknowledged with thanks.
Contents

Abstract i

Acknowledgement ii

Nomenclature viii

List of Figures xi

List of Tables xix

1 Introduction 1
  1.1 Motives for Research on Gas Turbine Engines 1
  1.2 Goals and Objectives 2
  1.3 Overview of Previous Research on Combustors 4
     1.3.1 Experiments on Combustor Flows 4
     1.3.2 Computations on Combustor Flows 6

2 Governing Equations and Mathematical Modeling 10
  2.1 Mean Flow Equations 10
  2.2 Two-Equation Model \( k-\varepsilon \) 13
  2.3 Chemistry Model 16
     2.3.1 Conservation of Chemical Species 16
     2.3.2 One-Step Overall Chemical Reaction 17
     2.3.3 Chemical Mean Formation Rate Models 18
  2.4 Conservation of Energy 20
4.2.5 Particle Seeding ................................................. 47
4.2.6 Signal Processing .............................................. 48
4.2.7 Data Reduction .................................................. 49
4.3 Flow Visualization ............................................... 50
4.4 Accuracy of Measurements ..................................... 51
  4.4.1 Control Volume ............................................... 51
  4.4.2 Statistical Bias ............................................... 51

5 Combustor Flow Without Fuel Injection .................. 54
  5.1 Flow Visualization ............................................ 54
  5.2 Experimental Results .......................................... 55
  5.3 Flow Velocities ................................................ 56
    5.3.1 Velocity Vectors ......................................... 56
    5.3.2 Total Velocity $V_t$ ..................................... 58
    5.3.3 Velocity $\bar{u}$ .......................................... 59
    5.3.4 Velocity $\bar{v}$ .......................................... 61
  5.4 Reynolds Stresses $\bar{u}^2$, $\bar{v}^2$ and $\bar{u}'\bar{v}'$ .. 62
    5.4.1 Normal Stress $\bar{u}^2$ .................................. 62
    5.4.2 Normal Stress $\bar{v}^2$ .................................. 64
    5.4.3 Shear Stress $\bar{u}'\bar{v}'$ ................................. 65
    5.4.4 Shear Stress Intensity SI .................................. 67
    5.4.5 Reynolds Stress Correlation Coefficient $C_{uv}$ ....... 69
    5.4.6 Ratio of the Normal Stress Intensities ................. 70
    5.4.7 Turbulence Kinetic Energy $k$ ............................. 71
    5.4.8 Ratio of $|\bar{u}'\bar{v}'|/k$ .................................. 73
    5.4.9 Turbulence Intensity TI ................................... 75
  5.5 Combustor Flow Structures in Cold Flow Without Fuel Injection .............................................. 77

6 Combustor Flow With Fuel Injection .......................... 80
  6.1 Flow Visualization ............................................ 80
  6.2 Experimental Results .......................................... 81
## Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>area [m$^2$]; pre-exponential rate constant in Arrhenius law</td>
</tr>
<tr>
<td>$C_1, C_2$</td>
<td>empirical constants in the dissipation equation</td>
</tr>
<tr>
<td>$C_D$</td>
<td>empirical constant in equation for turbulence kinetic energy</td>
</tr>
<tr>
<td>$C_p$</td>
<td>specific heat at constant pressure [J/kg·K]</td>
</tr>
<tr>
<td>$C_\mu$</td>
<td>empirical constant in equation for turbulence viscosity</td>
</tr>
<tr>
<td>$D$</td>
<td>diffusion coefficient [m$^2$/s]</td>
</tr>
<tr>
<td>$E$</td>
<td>integration constant in the law of the wall; activation energy [J/mole]</td>
</tr>
<tr>
<td>$H$</td>
<td>specific stagnation enthalpy [J/kg]</td>
</tr>
<tr>
<td>$J$</td>
<td>diffusion flux vector</td>
</tr>
<tr>
<td>$Le$</td>
<td>Lewis number = $Sc/Pr$</td>
</tr>
<tr>
<td>$M$</td>
<td>molecular weight of chemical species [kg/mole]</td>
</tr>
<tr>
<td>$N$</td>
<td>number of cycles</td>
</tr>
<tr>
<td>$P$</td>
<td>cell center; inlet pressure [Pa]</td>
</tr>
<tr>
<td>$Pe$</td>
<td>Peclet number</td>
</tr>
<tr>
<td>$R$</td>
<td>universal gas constant [J/mole·K]</td>
</tr>
<tr>
<td>$R$</td>
<td>chemical formation or reaction rate [kg/m$^3$·s]; residual source</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$S$</td>
<td>volumetric source term</td>
</tr>
<tr>
<td>$Sc$</td>
<td>Schmidt number = $\mu/\rho·D$</td>
</tr>
<tr>
<td>$Sp$</td>
<td>implicit source term</td>
</tr>
<tr>
<td>$Su$</td>
<td>explicit source term</td>
</tr>
<tr>
<td>$S1, S2$</td>
<td>empirical constants in laminar viscosity equation</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature [K], sampling time interval [s]</td>
</tr>
<tr>
<td>$U$</td>
<td>velocity in wall layer [m/s]</td>
</tr>
<tr>
<td>$V$</td>
<td>velocity [m/s]; volume [m$^3$]</td>
</tr>
<tr>
<td>$V_t$</td>
<td>characteristic velocity</td>
</tr>
<tr>
<td>$Y$</td>
<td>normal distant from wall boundary</td>
</tr>
<tr>
<td>$Y^+$</td>
<td>local Reynolds number = $\frac{\rho u_+ Y}{\mu}$</td>
</tr>
</tbody>
</table>
Greek characters

\begin{align*}
\alpha & \quad \text{relaxation factor} \\
\Gamma & \quad \text{effective exchange or diffusion coefficient \ [kg/m\cdot s]} \\
\delta & \quad \text{Kronecker tensor} \\
\delta_f & \quad \text{fringe spacing \ [m]} \\
\epsilon & \quad \text{dissipation rate of turbulence kinetic energy \ [m^2/s^3]} \\
\theta & \quad \text{intersection angle of laser beams \ [deg]} \\
\kappa & \quad \text{von Karman constant} \\
\lambda & \quad \text{thermal conductivity \ [N/s\cdot K] ; wavelength \ [nm]} \\
\mu & \quad \text{laminar viscosity \ [N\cdot s/m^2]} \\
\mu_t & \quad \text{turbulence viscosity \ [N\cdot s/m^2]} \\
\rho & \quad \text{density \ [kg/m^3]} \\
\sigma & \quad \text{standard deviation} \\
\sigma_\phi & \quad \text{Prandtl or Schmidt number for variable } \phi \\
\tau & \quad \text{shear stress}
\end{align*}
\( \phi \) general dependent variable

**Superscript**
- \( o \) reference state
- \( \rightarrow \) Reynolds averaged mean value
- \( \leftrightarrow \) vector
- \( \sim \) tensor
- \( \dot{\,} \) Favre averaged value; new calculated value
- \( \dot{\,} \) fluctuating component in Reynolds averaging; correction term
- \( \ddot{\,} \) fluctuating component in Favre averaging
- \( \ast \) guessed value

**Subscript**
- \( F \) fixed boundary value
- \( f \) false
- \( fu \) fuel
- \( eff \) effective
- \( H \) heat transfer
- \( i, j, k, \ell \) Cartesian tensor indices
- \( k \) chemical species
- \( N, S, E, W \) compass points for grid nodes
- \( n, s, e, w \) compass points for cell boundaries
- \( p \) particle
- \( s \) scattered light
- \( t \) total
- \( t \) turbulent
- \( w \) wall
List of Figures

1. Schematic diagram of staggered grids: (a) c cell; (b) u cell; (c) v cell.
2. Prescription of $\phi$ for western wall.
3. Boundary inlets: (a) north wall; (b) west wall.
4. Photograph of the sector rig.
5. Schematic diagram of the cross-section of the sector-combustor.
6. Schematic diagram of the air and fuel supply circuitry.
7. Principles of LDV: (a) Differential Doppler or dual-beam mode; (b) Fringe model.
8. Schematic diagram of LDV and data acquisition system.
9. Schematic diagram of seed generator.
10. Schematic diagram of the three sections investigated in cold flow without fuel injection.
11. Velocity vectors in Section 1 of the cold flow without fuel injection.
12. Velocity vectors in Section 2 of the cold flow without fuel injection.
13. Velocity vectors in Section 3 of the cold flow without fuel injection.
14. Contours of the total velocity $V_t$ in Section 1 of the cold flow without fuel injection.
15. Surface plot of the total velocity $V_t$ in Section 1 of the cold flow without fuel injection.
16. Contours of the total velocity $V_t$ in Section 2 of the cold flow without fuel injection.
17. Surface plot of the total velocity $V_t$ in Section 2 of the cold flow without fuel injection.
18. Contours of the total velocity $V_t$ in Section 3 of the cold flow without fuel injection.
19. Surface plot of the total velocity $V_t$ in Section 3 of the cold flow without fuel injection.
20. Contours of the longitudinal velocity $\bar{u}$ in Section 1 of the cold flow without fuel injection.
21. Surface plot of the longitudinal velocity $\bar{u}$ in Section 1 of the cold flow without fuel injection.

22. Contours of the longitudinal velocity $\bar{u}$ in Section 2 of the cold flow without fuel injection.

23. Surface plot of the longitudinal velocity $\bar{u}$ in Section 2 of the cold flow without fuel injection.

24. Contours of the longitudinal velocity $\bar{u}$ in Section 3 of the cold flow without fuel injection.

25. Surface plot of the longitudinal velocity $\bar{u}$ in Section 3 of the cold flow without fuel injection.

26. Contours of the transverse velocity $\bar{v}$ in Section 1 of the cold flow without fuel injection.

27. Surface plot of the transverse velocity $\bar{v}$ in Section 1 of the cold flow without fuel injection.

28. Contours of the transverse velocity $\bar{v}$ in Section 2 of the cold flow without fuel injection.

29. Surface plot of the transverse velocity $\bar{v}$ in Section 2 of the cold flow without fuel injection.

30. Contours of the transverse velocity $\bar{v}$ in Section 3 of the cold flow without fuel injection.

31. Surface plot of the transverse velocity $\bar{v}$ in Section 3 of the cold flow without fuel injection.

32. Contours of the normal stress $\bar{u}\bar{v}$ in Section 1 of the cold flow without fuel injection.

33. Surface plot of the normal stress $\bar{u}\bar{v}$ in Section 1 of the cold flow without fuel injection.

34. Contours of the normal stress $\bar{u}\bar{v}$ in Section 2 of the cold flow without fuel injection.

35. Surface plot of the normal stress $\bar{u}\bar{v}$ in Section 2 of the cold flow without fuel injection.

36. Contours of the normal stress $\bar{u}\bar{v}$ in Section 3 of the cold flow without fuel injection.

37. Surface plot of the normal stress $\bar{u}\bar{v}$ in Section 3 of the cold flow without fuel injection.

38. Contours of the normal stress $\bar{v}\bar{v}$ in Section 1 of the cold flow without fuel injection.

39. Surface plot of the normal stress $\bar{v}\bar{v}$ in Section 1 of the cold flow without fuel injection.

40. Contours of the normal stress $\bar{v}\bar{v}$ in Section 2 of the cold flow without fuel injection.

41. Surface plot of the normal stress $\bar{v}\bar{v}$ in Section 2 of the cold flow without fuel injection.

42. Contours of the normal stress $\bar{v}\bar{v}$ in Section 3 of the cold flow without fuel injection.

43. Surface plot of the normal stress $\bar{v}\bar{v}$ in Section 3 of the cold flow without fuel injection.

44. Contours of the shear stress $\bar{u}'\bar{v}'$ in Section 1 of the cold flow without fuel injection.

45. Surface plot of the shear stress $\bar{u}'\bar{v}'$ in Section 1 of the cold flow without fuel injection.

46. Contours of the shear stress $\bar{u}'\bar{v}'$ in Section 2 of the cold flow without fuel injection.

47. Surface plot of the shear stress $\bar{u}'\bar{v}'$ in Section 2 of the cold flow without fuel injection.

48. Contours of the shear stress $\bar{u}'\bar{v}'$ in Section 3 of the cold flow without fuel injection.
49. Surface plot of the shear stress $u'v'$ in Section 3 of the cold flow without fuel injection.
50. Contours of the shear stress intensity in Section 1 of the cold flow without fuel injection.
51. Contours of the shear stress intensity in Section 2 of the cold flow without fuel injection.
52. Contours of the shear stress intensity in Section 3 of the cold flow without fuel injection.
53. Contours of the Reynolds stress correlation coefficient in Section 1 of the cold flow without fuel injection.
54. Contours of the Reynolds stress correlation coefficient in Section 2 of the cold flow without fuel injection.
55. Contours of the Reynolds stress correlation coefficient in Section 3 of the cold flow without fuel injection.
56. Contours of the ratio of the normal stress intensities in Section 1 of the cold flow without fuel injection.
57. Contours of the ratio of the normal stress intensities in Section 2 of the cold flow without fuel injection.
58. Contours of the ratio of the normal stress intensities in Section 3 of the cold flow without fuel injection.
59. Contours of the turbulence kinetic energy $k$ in Section 1 of the cold flow without fuel injection.
60. Surface plot of the turbulence kinetic energy $k$ in Section 1 of the cold flow without fuel injection.
61. Contours of the turbulence kinetic energy $k$ in Section 2 of the cold flow without fuel injection.
62. Surface plot of the turbulence kinetic energy $k$ in Section 2 of the cold flow without fuel injection.
63. Contours of the turbulence kinetic energy $k$ in Section 3 of the cold flow without fuel injection.
64. Surface plot of the turbulence kinetic energy $k$ in Section 3 of the cold flow without fuel injection.
65. Contours of the ratio of $|u'v'|/k$ in Section 1 of the cold flow without fuel injection.
66. Contours of the ratio of $|u'v'|/k$ in Section 2 of the cold flow without fuel injection.
67. Contours of the ratio of $|u'v'|/k$ in Section 3 of the cold flow without fuel injection.
68. Contours of the turbulence intensity in Section 1 of the cold flow without fuel injection.
69. Contours of the turbulence intensity in Section 2 of the cold flow without fuel injection.
70. Contours of the turbulence intensity in Section 3 of the cold flow without fuel injection.
71. Streaklines in Section 1 of the cold flow without fuel injection.
72. Streaklines in Section 2 of the cold flow without fuel injection.
73. Streaklines in Section 3 of the cold flow without fuel injection.
74. Photographs of the combustion flame: (a) side view; b) rear view.
75. Schematic diagram of the three sections investigated in the cold and hot flows with fuel injection.
76. (a) Velocity vector plot in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
76. (b) Velocity vector plot in Section 1 (Fuel-Injector Plane) of the hot flow.
77. Velocity vector plot in Section 2 of the hot flow.
78. Velocity vector plot in Section 3 of the hot flow.
79. The contour plot of the total velocity $V_t$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
80. The surface plot of the total velocity $V_t$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
81. The contour plot of the total velocity $V_t$ in Section 1 (Fuel-Injector Plane) of the hot flow.
82. The surface plot of the total velocity $V_t$ in Section 1 (Fuel-Injector Plane) of the hot flow.
83. The contour plot of the total velocity $V_t$ in Section 2 of the hot flow.
84. The surface plot of the total velocity $V_t$ in Section 2 of the hot flow.
85. The contour plot of the total velocity $V_t$ in Section 3 of the hot flow.
86. The surface plot of the total velocity $V_t$ in Section 3 of the hot flow.
87. The contour plot of the longitudinal velocity $\bar{u}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
88. The surface plot of the longitudinal velocity $\bar{u}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
89. The contour plot of the longitudinal velocity $\bar{u}$ in Section 1 (Fuel-Injector Plane) of the hot flow.
90. The surface plot of the longitudinal velocity $\bar{u}$ in Section 1 (Fuel-Injector Plane) of the hot flow.
91. The contour plot of the longitudinal velocity $\bar{u}$ in Section 2 of the hot flow.
92. The surface plot of the longitudinal velocity $\bar{u}$ in Section 2 of the hot flow.
93. The contour plot of the longitudinal velocity $\bar{u}$ in Section 3 of the hot flow.
94. The surface plot of the longitudinal velocity $\bar{u}$ in Section 3 of the hot flow.
95. The contour plot of the transverse velocity $\bar{v}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

96. The surface plot of the transverse velocity $\bar{v}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

97. The contour plot of the transverse velocity $\bar{v}$ in Section 1 (Fuel-Injector Plane) of the hot flow.

98. The surface plot of the transverse velocity $\bar{v}$ in Section 1 (Fuel-Injector Plane) of the hot flow.

99. The contour plot of the transverse velocity $\bar{v}$ in Section 2 of the hot flow.

100. The surface plot of the transverse velocity $\bar{v}$ in Section 2 of the hot flow.

101. The contour plot of the transverse velocity $\bar{v}$ in Section 3 of the hot flow.

102. The surface plot of the transverse velocity $\bar{v}$ in Section 3 of the hot flow.

103. The contour plot of the normal stress $\bar{u}^{2}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

104. The surface plot of the normal stress $\bar{u}^{2}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

105. The contour plot of the normal stress $\bar{u}^{2}$ in Section 1 (Fuel-Injector Plane) of the hot flow.

106. The surface plot of the normal stress $\bar{u}^{2}$ in Section 1 (Fuel-Injector Plane) of the hot flow.

107. The contour plot of the normal stress $\bar{u}^{2}$ in Section 2 of the hot flow.

108. The surface plot of the normal stress $\bar{u}^{2}$ in Section 2 of the hot flow.

109. The contour plot of the normal stress $\bar{u}^{2}$ in Section 3 of the hot flow.

110. The surface plot of the normal stress $\bar{u}^{2}$ in Section 3 of the hot flow.

111. The contour plot of the normal stress $\bar{v}^{2}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

112. The surface plot of the normal stress $\bar{v}^{2}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

113. The contour plot of the normal stress $\bar{v}^{2}$ in Section 1 (Fuel-Injector Plane) of the hot flow.

114. The surface plot of the normal stress $\bar{v}^{2}$ in Section 1 (Fuel-Injector Plane) of the hot flow.

115. The contour plot of the normal stress $\bar{v}^{2}$ in Section 2 of the hot flow.

116. The surface plot of the normal stress $\bar{v}^{2}$ in Section 2 of the hot flow.

117. The contour plot of the normal stress $\bar{v}^{2}$ in Section 3 of the hot flow.
118. The surface plot of the normal stress $\overline{v'^2}$ in Section 3 of the hot flow.

119. The contour plot of the shear stress $\overline{u'v'}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

120. The surface plot of the shear stress $\overline{u'v'}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

121. The contour plot of the shear stress $\overline{u'v'}$ in Section 1 (Fuel-Injector Plane) of the hot flow.

122. The surface plot of the shear stress $\overline{u'v'}$ in Section 1 (Fuel-Injector Plane) of the hot flow.

123. The contour plot of the shear stress $\overline{u'v'}$ in Section 2 of the hot flow.

124. The surface plot of the shear stress $\overline{u'v'}$ in Section 2 of the hot flow.

125. The contour plot of the shear stress $\overline{u'v'}$ in Section 3 of the hot flow.

126. The surface plot of the shear stress $\overline{u'v'}$ in Section 3 of the hot flow.

127. The contour plot of the shear stress intensity in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

128. The contour plot of the shear stress intensity in Section 1 (Fuel-Injector Plane) of the cold flow.

129. The contour plot of the shear stress intensity in Section 2 of the hot flow.

130. The contour plot of the shear stress intensity in Section 3 of the hot flow.

131. The contour plot of the Reynolds stress correlation coefficient in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

132. The contour plot of the Reynolds stress correlation coefficient in Section 1 (Fuel-Injector Plane) of the hot flow.

133. The contour plot of the Reynolds stress correlation coefficient in Section 2 of the hot flow.

134. The contour plot of the Reynolds stress correlation coefficient in Section 3 of the hot flow.

135. The contour plot of the ratio of the normal stress intensities in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

136. The contour plot of the ratio of the normal stress intensities in Section 1 (Fuel-Injector Plane) of the hot flow with fuel injection.

137. The contour plot of the ratio of the normal stress intensities in Section 2 of the hot flow.

138. The contour plot of the ratio of the normal stress intensities in Section 3 of the hot flow.

139. The contour plot of the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

140. The surface plot of the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
141. The contour plot of the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the hot flow.

142. The surface plot of the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the hot flow.

143. The contour plot of the turbulence kinetic energy $k$ in Section 2 of the hot flow.

144. The surface plot of the turbulence kinetic energy $k$ in Section 2 of the hot flow.

145. The contour plot of the turbulence kinetic energy $k$ in Section 3 of the hot flow.

146. The surface plot of the turbulence kinetic energy $k$ in Section 3 of the hot flow.

147. The contour plot of the ratio of the shear stress $u'v'$ and the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

148. The contour plot of the ratio of the shear stress $u'v'$ and the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the hot flow.

149. The contour plot of the ratio of the shear stress $u'v'$ and the turbulence kinetic energy $k$ in Section 2 of the hot flow.

150. The contour plot of the ratio of the shear stress $u'v'$ and the turbulence kinetic energy $k$ in Section 3 of the hot flow.

151. The contour plot of the turbulence intensity in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

152. The contour plot of the turbulence intensity in Section 1 (Fuel-Injector Plane) of the hot flow.

153. The contour plot of the turbulence intensity in Section 2 of the hot flow.

154. The contour plot of the turbulence intensity in Section 3 of the hot flow.

155. Comparison of the mean velocity $\bar{u}$ between the hot and cold flows.

156. Comparison of the transverse mean velocity $\bar{v}$ between the hot and cold flows.

157. Comparison of the turbulence kinetic energy $k$ between the hot and cold flows.

158. Comparison of the shear stress $u'v'$ between the hot and cold flows.

159. Streaklines in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.

160. Streaklines in Section 1 (Fuel-Injector Plane) of the hot flow.

161. Streaklines in Section 2 (Fuel-Injector Plane) of the hot flow.

162. Streaklines in Section 3 (Fuel-Injector Plane) of the hot flow.

163. The computed streamline in Section 1 (Top-Jet Plane) of the cold flow without fuel injection.

164. The computed streamline in Section 2 (Inter-Jets Plane) of the cold flow without fuel injection.
165. The computed streamline in Section 3 (Bottom-Jet Plane) of the cold flow without fuel injection.

166. The computed streamline in the Hypothetical Plane of the cold flow without fuel injection.

167. Comparison of the measured and the predicted longitudinal velocity \( \bar{u} \) for cold flow without fuel injection.

168. Comparison of the measured and the predicted transverse velocity \( \bar{v} \) for cold flow without fuel injection.

169. Comparison of the measured and the predicted turbulence kinetic energy \( k \) for cold flow without fuel injection.

170. Comparison of the measured and the predicted shear stress \( \bar{u}'\bar{v}' \) for cold flow without fuel injection.

171. The computed streamline in Section 1 (Top-Jet Plane) of the cold flow with fuel injection.

172. The total velocity \( V_t \) of the predicted flow in Section 1 (Top-Jet Plane) of the cold flow with fuel injection.

173. The mixture fraction \( f \) of the predicted flow in Section 1 (Top-Jet Plane) of the cold flow with fuel injection.

174. Velocity vectors in Section 1 measured by LDV and predicted by Lai (1990) in cold flow without fuel injection.

175. Velocity vectors in Section 2 measured by LDV and predicted by Lai (1990) in cold flow without fuel injection.

176. Velocity vectors in Section 3 measured by LDV and predicted by Lai (1990) in cold flow without fuel injection.
List of Tables

1. Constants in $k - \varepsilon$ model.
2. Entries of $\phi$ and $R_k$.
Chapter 1

Introduction

1.1 Motives for Research on Gas Turbine Engines

Gas turbine engines have successfully been used in the aviation industry for many decades. The power of such engines is produced by the release of chemical energy from the fuel through turbulent mixing and burning of the air-fuel mass inside the combustor. Due to the complexity of the flowfield in these combustors, their aerodynamic and physical geometry designs have been based on past experience, empirical formulae, and extensive, time-consuming and costly trial and error developmental testings.

Since the last decade, very serious research and development activities have been directed towards a better understanding and deeper knowledge of the complex flowfields in combustors. The reasons may be threefold. First, since the global energy crisis of the nineteen seventies, there has been increasing concern for fuel economy; combustion devices had to be more efficient thermodynamically. At the same time, more stringent restrictions were imposed on their pollutant emissions due to their effect on the environment. Second, the combustor, along with the turbine blades, plays a major role in the engine maintenance cost due to the hostile environment created for the engine components in the hot section. With advanced materials and cooling concepts, a significant increase in turbine inlet temperature becomes possible. Moreover, with the application of aerodynamics to the design of compressors, high pressure ratio compressors with adequate efficiency become a reality. Consequently, the performance of aircraft turbine engines improves significantly. However, the higher temperature and pressure ratio with corresponding high temperature cooling air have thus created a hostile environment for the engine components. Third, recent advances in diagnostic techniques and in the capabilities of digital computers have encouraged the mathematical and numerical modeling of complicated turbulent combusting flows. Non-intrusive diagnostic tools using light-scattering and optical techniques are now well established to reveal qualitative and quantitative information in turbulent and combusting flows. Laser-induced fluorescence (LIF), laser Rayleigh and Raman scattering techniques, and coherent anti-Stokes Raman spectroscopy (CARS) are used for measuring the concentration of chemical species, density and temperature, and planar and three-dimensional imagings. The well-known particle Mie scattering process is used in laser
Doppler velocimetry (LDV). LDV enables the measurement of instantaneous fluctuating flow velocities which was quite impossible with other techniques in many hostile environments. The LDV measurements of many statistical properties of the velocity field thus create an extensive and detailed bench-mark data base which is crucial to the mathematical modeling of turbulent flows. The combination of LDV with LIF, Rayleigh scattering, Raman scattering, or CARS enables simultaneous velocity and scalar measurements, and subsequently provides spectral information and joint probability density distributions of velocity and scalar properties and their correlations. For fuel spray studies, the particle measurement system (PMS), which is based on laser spectrometry, permits rapid spray analysis for a broad spectrum of particle diameters, and the phase Doppler particle analyzer (PDPA) is capable of measuring the size distribution of droplets, their velocity, flux and concentration. The tremendous improvement in the speed and storage capacity of digital computers allows more extensive and complicated numerical codes to be developed for turbulent combusting flows. Complex turbulence models can now be implemented and tested for their applicability and universality. Many two-dimensional codes may also be extended to three dimensions as the computer storage increases and cost decreases.

The development of gas turbine combustors has always been expensive and very involved with the testing of new configurations in full-scale rigs. Diagnostic testing in sector rigs can be setup and operated with greater ease and lower cost and are favored for preliminary studies. If the combustion characteristics, aerodynamics and flow properties can be predicted and optimized a priori, the costs of developing, running and maintaining the engine would be reduced drastically. Computational fluid dynamic codes have shown great potential in becoming a valuable design tool for application to the confined internal flow of a gas turbine engine.

For helicopter and turboprop engines with front power shafts, the normal practice is to locate the turbine inside under the combustor so as to shorten the length of the power shaft. Such design compelled the reverse-flow annular configuration rather than the straight-through flow configuration.

1.2 Goals and Objectives

Although a number of experimental investigations were performed to understand flows in combustors, there exists a deficiency in detailed information on the velocity characteristics within real combustors. From the modeling point of view, failure to accurately predict the aerodynamic flowfield has precluded the complete testing of combustion models. Therefore, reliable velocity field measurements within combustors become the current priority and requirement before improvements in the evaluation of the existing turbulence and combustion models for real combustors can be achieved. Therefore, there is a strong demand and necessity of bench-mark quality experimental data on a practical gas-turbine combustor flow from the design, analytical and numerical modeling points of view.
The annular combustion chamber has been extensively used in gas turbine engines. Its main advantage over the multi-combustion chamber and the tubo-annular combustion chamber is that it utilizes the space allocated to the combustor in the layout of the engine efficiently. With the same performance level and diameter, the mass of an annular combustor is merely 75% that of the tubo-annular combustor, hence reducing the weight of the engine drastically and lowering the cost of manufacturing such engines. Another advantage of this type of engine is its better flame propagation characteristics since it does not require any connection tube between chambers for the flame to propagate. The objectives of the present study are:

- to obtain detailed mapping of the two mean velocity components, the corresponding normal stresses and shear stress in a Pratt and Whitney Canada toroidal-vortex annular reverse-flow sector-combustor using the laser Doppler velocimetry technique (LDV),
- to identify the flowfield characteristics under cold flow without fuel injection, and cold and hot flows with fuel injection conditions,
- to determine from the measurements the effects of combustion on the flowfield due to heat addition,
- to develop a two-dimensional computer code to study steady-state, turbulent, confined, recirculating, compressible, non-reactive and reactive flow,
- to validate the applicability of the two-dimensional computer code in predicting combustor flowfield,
- to validate the applicability of the two-equation $k-\varepsilon$ turbulence model in predicting complex combustor flow.

The present research effort strives toward the study of a real, practical annular combustor with a sector arrangement which permits ready access to the entire flowfield of the combustor geometry. The collected measurements present an important data base for combustor researchers, combustor flow modelers and numerical analysts.

Pratt and Whitney Canada provided a sector rig of a toroidal-vortex annular reverse-flow combustor of the PW 200 series engine for the experimental investigation. Measurements were obtained with a two-component Argon-ion laser Doppler velocimetry system. For the cold flow experiments, no fuel injector nor ignitor were inserted into the rig. The purpose of the cold flow measurements without fuel injection was to obtain and understand the aerodynamics of the combustor design, to identify the flow characteristics, and to extract quantitative information on the flowfield. The experiments of the cold flow with fuel injection would show the changes in the flowfield due to the addition of a fuel jet in the primary zone. Hot flow measurements would then provide a real picture of the conditions in a real engine during its operation. A comparison of the cold flow and hot flow data in the same physical plane and under the same operating conditions would allow the determination of the effects of heat addition to the flowfield due to combustion. Main contributions from the experimental investigation include the study of a sector rig whose geometrical details are of a real-size, practical annular combustor, and whose
design is in accord with the most current state-of-the-art gas turbine engine developments; measurements of the \( u \) and \( v \) components of the mean velocity, the corresponding normal stresses \( u'^2 \) and \( v'^2 \), and the shear stress \( u'v' \); detailed data acquisition of the above flow properties on several sections of the flow domain. It should be emphasized here that such detailed mapping of the flow properties inside a practical combustor is believed to be the first effort ever attempted in the field of gas turbine engine research.

Among the various theoretical turbulence closure models (see, for example, Launder and Spalding 1972), the two-equation \( k-\varepsilon \) turbulence model (Jones and Launder 1972) has been the most widely used and successfully applied model in many practical problems. It utilizes two quantities in characterizing turbulence and in determining the eddy diffusivity. A nonlinear differential equation is obtained for each quantity; therefore, it involves less empiricism than the one-equation turbulence models or the algebraic mixing length models. The model requires no previous knowledge of the flow under consideration except the inlet and boundary conditions. Its simplicity and versatility has gained popularity in general engineering applications. A comparison of the computational results with the LDV measurements would indicate the applicability of the \( k-\varepsilon \) turbulence model in complex combustor flows.

The developed UTCM-2 (University of Toronto Combustor Modeling – 2 dimensional) computer code is based on the TEACH family of programs (Gosman and Ideriah 1978). The code is capable of solving two-dimensional planar and axisymmetric flows with complex physical geometry. It is very versatile and can be easily modified and extended. Some major features of this contribution include the development of a scheme for treating top, bottom, forward-facing and backward-facing boundaries; development of a scheme for treating free jets and wall jets at their inlets; incorporation of the SIMPLEC algorithm (Van Doormal and Raithby 1984) in solving the Poisson equation for the pressure field; implementation of the energy equation for the computation of compressible and reactive flows; addition of a combustion model in which two more partial differential equations have to be solved; incorporation of an efficient algorithm for the determination of temperature in a multiple chemical species flow; extension of the Tri-Diagonal Matrix Algorithm (TDMA) to march alternately in two directions; the control parameters (such as the number of iterations, line-by-line marching directions, choice of compressible or incompressible, reactive or non-reactive, under-relaxation factors, geometrical boundaries and inlet port conditions) are stored in a separate file outside the code; and lastly, the under-relaxation factors and the number of iterations could be modified while executing the program in the Gould 32/9705 computer system. Besides the code, several plotting programs were developed to illustrate the computational results and experimental data in two-dimensional and three-dimensional fashions.

1.3 Overview of Previous Research on Combustors

1.3.1 Experiments on Combustor Flows

Experimental measurements of flow properties in real combustor geometries of gas turbine engines have been very limited and a bench-mark data base for a practical combustor is not
yet found in the literature. This is due to the cost, difficulty, and complexity of reaching all the flow regimes with probes or optical instrumentation, as the sector arrangement is a real combustor geometry. Because of the importance of the knowledge of combustor aerodynamics in combustor design, experiments have been performed for isothermal and combusting flows in simple model combustors. These simplified geometries were specially designed to allow the investigation of particular variables in the models relevant to real combustors.

In the study of isothermal flows in simplified combustor geometries, experiments have been performed in the primary, dilution, and the combined primary and dilution zones. Faler and Leibovich (1978) have determined the internal structure of a recirculating region using LDV. The domain and dependence of the recirculating region on the degree of swirl was explored by Escudier and Zender (1982) using LDV. Swirling flow in the primary zone was examined with LDV by Altgeld, Jones and Wilhelmi (1983). The study of the secondary zone was reported by Crabb and Whitelaw (1979) and Khan and Whitelaw (1980 a), using probe techniques, by simulating the interaction of a row of dilution jets in a crossflow. More realistic simulation of the dilution zone was done by extending the single row of jets to opposing rows of jets discharging into a cross-stream as described by Khan and Whitelaw (1980 b), Atkinson, Khan and Whitelaw (1983), and Wittig, Elbahar and Noll (1984). In the combined study of the primary and dilution zones, Green and Whitelaw (1980 and 1983) used LDV to measure flow velocities and turbulence intensities for turbulent isothermal flows in a water model with axisymmetric geometries related to gas-turbine combustors. They quantified the effects of the change from a two-dimensional to a three-dimensional geometry. Their measurements were compared with computed results which employed a two-equation turbulence model. Recently, the influence of primary holes on combustion efficiency was reported by Cadiou and Griene (1989). They found that the primary jets in a reverse flow combustor decreased the exhaust gas temperature without much influence on the combustor efficiency. Carrotte and Stevens (1989) study the influence of dilution hole geometry in crossflow. They designed a new dilution hole geometry that would reduce the skewness of the temperature distributions in dilution jets.

Swirling flows find an important application in many combustion chambers by providing flame stability and improved mixing (Syred and Beer 1974). Lilley and co-workers (1985) conducted experimental and theoretical research on axisymmetric geometries typical of gas turbine and ramjet combustion chambers. They studied the effects of a complete range of swirl strengths, swirler performance, downstream contraction nozzle sizes and locations, expansion ratios, and inlet side-wall angles on the flowfield. Their experimental work included flow visualization, pitot probe and hot-wire anemometry measurements of velocities and turbulence stresses. Measured data was compared with computed results. Authors who have conducted experiments of swirling jets in model combustors include Yetter and Gouldin (1976), Vu and Gouldin (1980), Habib and Whitelaw (1980), Ramos and Sommer (1984).

The influences of fuel-air ratio, inlet-air temperature, combustor pressure, and swirl on the combusting flowfield were examined by Owen, Spadaccini and Bowman (1977), Owen, Spadaccini, Kennedy and Bowman (1979), Nicholls et al. (1980), El Banhawy and Whitelaw (1981), Brum and Samuelsen (1982) and Attiya and Whitelaw (1984). In general, an increase in any of these parameters results in an increase of combustion efficiency and a decrease in CO and UHC levels.
The aforementioned investigations provided useful information on flow properties, which is essential to a better understanding of combustor flows; however, they were limited by the geometry of the combustor which was simpler than gas-turbine combustor geometries used in practice. Hence they cannot represent the flow properties of real combustors entirely where the intense mixing and combusting processes in the primary zone is truly three-dimensional in nature, and the trajectory of the dilution jets can be strongly related to the combustor geometry. Even with simple geometries where access to the flow regime is readily attainable, several authors including Bilger (1977), Attya and Whitelaw (1984), and LaRue, Samuelsen and Seiler (1984) have reported that accurate measurement of any flow properties is difficult.

From the beginning of the eighties, some research efforts have begun towards the investigation on models of real combustor geometries and sector rigs. Toral and Whitelaw (1982) obtained measurements in an isothermal and combusting flow of a sector of an annular combustor. LDV, thermocouples, pitot probe, water-cooled probe and gas detector and analyzers were used to measure components of mean velocity, mean temperature, and concentrations of chemical species. However, only the mean axial and transverse velocities at limited locations were reported. Goldstein, Lau and Leung (1983) obtained LDV data for the mean and fluctuating axial velocity profile but it was restricted to the plane lying downstream of their combustor exit. Kraemer (1985) used LDV to examine the flow field within an AVCO-Lycoming PLT-34 annular combustor liner under cold flow conditions. The circumferentially-stirred combustor design produces a highly three-dimensional flowfield without a swirler. However, only the mean and root mean square velocity profiles were measured at limited locations in the primary zone. Heitor (1985) and Heitor and Whitelaw (1986) measured the velocity, temperature, and species in a can-type combustor with isothermal and combusting flows. They studied the influence of combustion, air-fuel ratio, and preheated inlet air temperature on the flowfield. The mean axial and tangential velocities and the Reynolds stresses obtained with LDV were limited to sections of the primary and dilution holes, and the exit plane. Recently, Bicen, Tse and Whitelaw (1987) studied the flow and combustion characteristics of an annular Gem-60 sector-combustor. They measured the mean longitudinal velocity and their corresponding normal stress by LDV, temperature by fine wire thermocouples, and concentrations of chemical species through microprobes. Again, measurements were limited only to the primary zone and the exit plane.

Quite complete compilations of experimental data base on element test related to combustor internal flows are given by Srinivasan et al. (1983), Kenworthy, Correa and Burrus (1983) and Sturgess (1983). Reviews of recent experimental works on simple model combustors and gas turbine combustors are reported by Heitor (1989).

1.3.2 Computations on Combustor Flows

Computational fluid dynamics (CFD) is now being applied to propulsion systems and to gas turbine engines in order to reduce the development cost and time of aerospace propulsion systems. Simulation of flows within a gas turbine engine using CFD has been lagging behind the simulation of airframe aerodynamics due to the inherently complex flow regimes in in-
ternal flows. A well-known generic computer code originated at Imperial College as a CFD teaching tool is TEACH (acronym for Teaching Elliptic Axisymmetric Characteristic Heuristically) by Gosman and Ideriah (1983). The code is for isothermal, turbulent, recirculating, planar and axisymmetric flows with $k-\varepsilon$ turbulence model, wall functions and the SIMPLE algorithm (Patankar and Spalding 1972a, 1972b) implemented. Many researchers and scientists have adopted the TEACH code and expanded and modified it extensively under various new acronyms such as STARPIC (Lilley and Rhode 1982), FLUENT\(^1\), PREACH\(^2\), PACE (Coupland and Priddin 1986), INTERN (NREE 1981), TEMA (Lai and Salcudean 1985) and TURCOM (Lai 1987). Sturgess, James and Syed (1985) covered the scope of the computational problem in the calculation of gas turbine engine combustor flows. The state-of-the-art in such calculations was given and the balance between solution realism and accuracy with solution cost and machine demand was established. Green and Whitelaw (1980) used a two-dimensional computer code with $k-\varepsilon$ turbulence closure model to calculate the axisymmetric flow inside an annular combustor model. The calculated and measured results showed good agreement, but the discrepancies were found to depend more on the distribution of grid nodes than on the turbulence model. The turbulence modeling errors were prevalent in regions of recirculation, flow development and in the near-wall region. Lilley (1985) reviewed his research effort in establishing an improved simulation in the form of a computer prediction code equipped with a suitable turbulence model for predicting swirling recirculating confined turbulent flows. The developed STARPIC code (Lilley and Rhode 1982) is basically a TEACH program that has been appropriately modified to include swirl.

Chiappetta (1983) presented a description of the mathematical basis for, and operation of, a new version of the TEACH computer program which was supplied to United Technologies Research Center by Professor A.D. Gosman and his colleagues at Imperial College, London. The revised code was intended for the analysis of a cylindrical bluff-body research combustor developed by Rocquemore and colleagues at AFAPL as a research tool for gas turbine combustor modeling and for diagnostic instrumentation development (Rocquemore et al. 1980). The new version of the TEACH code is capable of solving two-dimensional subsonic, swirling, non-reacting and reacting, turbulent flow. It solves the elliptic steady-state equations of motion using an improved finite difference procedure, viz. The Bounded Skew-Upward Differencing (BSUD) method. The pressure field is estimated by means of a new algorithm — the Pressure-Implicit Split Operation (PISO) (Issa 1982) predictor-corrector technique. The rate of combustion of the fuel is represented by the eddy-dissipation combustion model proposed by Magnussen and Hjertager (1978). Nikjooy and So (1987) conducted a numerical study of non-reactive and reactive axisymmetric combustor flows with and without swirl. Closure of the Reynolds equations by three different turbulence models: $k-\varepsilon$ (Jones and Lauder 1972), algebraic stress (Mellor and Yamada 1974, Rodi 1976, Gibson and Launder 1976), and Reynolds stress (Hanjalić and Launder 1972a, 1972b) were employed in their studies. They analyzed the performance of several locally non-equilibrium and equilibrium algebraic stress models, and compared the results of high and low Reynolds number model for combustor flows using Reynolds stress closures. Two different models for the scalar transport were presented. Fast- and finite-rate chemistry models were applied to non-premixed combustion; however the

---

\(^1\) A trademark of Creare Research and Development, Inc.

\(^2\) A trademark of United Technologies Corporation, Pratt and Whitney Engineering Division.
finite-rate chemistry model was preferred for determining the combustion effects in combustor.

Nallasamy (1985) provided a critical evaluation of various turbulence models applicable to confined flows, and Syed, Chiappetta and Gosman (1985) reported the details of evaluating the best available finite difference scheme to reduce numerical error in combustor performance computer codes.

Computer codes which provide three-dimensional predictions of flow properties are relatively new. The methodology, structure and application of three-dimensional codes to elliptic flows have been reported by Patankar (1975, 1980) and Lai (1987). Applications of three-dimensional codes to combustor flows have been reported in literature by several authors. Busnaina (1985) investigated the primary and dilution lateral jet injections into a typical isothermal combustor. Green and Whitelaw (1983) simulated the flow properties inside two gas-turbine model combustors under isothermal flow conditions. Yan, Tang and Zhu (1985) predicted the flowfield inside an annular vaporizing combustor model under isothermal conditions. Kraemer (1985) predicted the horseshoe-shaped flowfield within an AVCO-Lycoming PLT-34 annular combustor using the INTERN code.

The ultimate goal of all current numerical modeling efforts is to validate and apply the modeling methods to a production gas turbine combustor, so that the flowfield properties, exit temperature pattern and emissions can be predicted during its development prior to manufacture and test. At the beginning of this decade, the Lewis Research Center of the National Aeronautics and Space Administration (NASA) initiated a large research project entitled Turbine Engine Hot Section Technology (HOST). This was a competitive, multi-phase program. As part of this effort, the aim of the Aerothermal Modeling Program was to provide a quantitatively accurate numerical modeling capability for the design and development of gas turbine engine combustors. The first phase was to evaluate the performance of current codes for internal flows. The study covered a wide range of flows which represent elements of a practical combustor. Final reports for the first phase were prepared by three contractors — Garrett Turbine Engine Co. (Srinivasan et al. 1983), General Electric Co. (Kenworthy, Correa and Burrus 1983), and Pratt and Whitney Aircraft Group (Sturgess 1983). They all reached similar conclusions on the performance of the TEACH-type codes evaluated in that the codes presently give qualitative agreement for combustor flows. However, considerable work remains to be done before the computational fluid dynamics codes can be recognized as viable design and development tools.

Although the subject of pollutant emission is beyond the scope of the present study, there has been a considerable amount of interest in the study of pollutant emission due to the new proposed jet aircraft emissions standards and the increasing concern, on a global scale, of the greenhouse effect. The following are examples of some of the research work done on pollutant emission. The importance of chemical kinetics and turbulent mixing on the controlling of emissions from gas turbines was studied by Gouldin (1973); Tuttle, Altenkirch and Mellor (1973) investigated the effect of inlet air temperature on the emissions from and within an Allison J-33 combustor; Owen, Spadaccini and Bowman (1976) investigated the effects of inlet air swirl, pressure and fuel/air velocity on pollutant formation and energy release in a confined turbulent diffusion flame combustor; Nicholls et al. (1980) studied the fuel
spray characteristics on emissions from a research gas turbine combustor; Danjo, Sawada and Nishi (1980) measured and analyzed the pollutant emissions from a reversed flow combustor during its accelerating and decelerating phases; Noyce, Sheppard and Yamba (1981) measured and analyzed the degree of fuel-air mixing on pollutant formation at idling conditions of a single can-type combustor; Noyce and Sheppard (1982) also examined the influence of varying equivalence ratio on pollutant formation in a single can-type combustor; Jones and Toral (1983) investigated the influence of air inlet temperature on the temperature profile and chemical species concentrations within a can-type model combustor.
Chapter 2

Governing Equations and Mathematical Modeling

In this chapter, the theoretical framework and the equations which govern the distribution of the mean flow quantities are presented. The turbulence closure scheme via the $k$-$\varepsilon$ model, their transport equations for high Reynolds number flows and the wall function treatment for near-wall regions are introduced. The formulations of the energy equation and the combustion model are also discussed.

2.1 Mean Flow Equations

In principle, the Navier-Stokes equations describe laminar as well as turbulent flows. Unfortunately, they cannot be used at present for turbulent flows because important details of turbulence are small-scale in nature. In order to resolve these small-scale motions, the grid size of the numerical scheme would have to be even finer. Therefore, the corresponding amount of storage for flow variables at so many grid points and the amount of computational time and cost become prohibitive. Fortunately, one is usually concerned only with the time-averaged effects of turbulence, even if the mean flow is unsteady. Hence, one bases predictions of turbulent flows only on time-averaged properties of turbulence.

The governing equations describing the instantaneous properties of a flow are the conservation equations of mass and momentum. In the absence of external force fields and compressibility effects, they can be expressed in Cartesian tensor notation (Bird, Stewart and Lightfoot, 1960) as:

mass conservation
\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \]  

(2.1)

momentum conservation
\[ \frac{\partial (\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \]  

(2.2)
where $u_i$ is the component of instantaneous velocity in the $x_i$ direction, $p$ is the instantaneous static pressure, and $\tau_{ij}$ is the instantaneous shear stress tensor.

\[
\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial u_\ell}{\partial x_\ell} \delta_{ij}
\] (2.3)

Here $\mu$ is the molecular viscosity and $\delta_{ij}$ is the Kronecker symbol. For incompressible flow, the second term in Eq. (2.3) will disappear for $\frac{\partial u_\ell}{\partial x_\ell} = 0$. Since it is quite impossible to track the behavior of the instantaneous flowfield analytically, and it is beyond the computing capability to provide a description of the instantaneous flowfield, the method used to circumvent these difficulties is to predict the mean flowfield which is statistically time or ensemble averaged. This approach was first proposed in the late 19th century by Osborne Reynolds. Dependent variables of turbulent flows are decomposed into statistical averages and random fluctuations. The statistical average of the instantaneous velocity $u_i$ with respect to time at a given point is given by

\[
\bar{u}_i = \lim_{T \to \infty} \frac{1}{2T} \int_{-T}^{T} u_i \, dt
\] (2.4)

where $T$ is a time interval long compared with the largest time scales, but shorter than the period over which the averaged flow quantities may vary. The instantaneous velocity can then be written as

\[
u_i = \bar{u}_i + u'_i
\] (2.5)

where $u'_i$ is the fluctuating velocity component. Taking the average of both sides of (2.5) gives the mean of the fluctuating component, which is identically zero.

\[
\bar{u}_i' = 0
\] (2.6)

This averaging procedure of the flow variables is mainly used for constant density or incompressible flows. For variable density flows, the density-weighted decomposition and averaging of a flow variable such as the instantaneous velocity $u_i$ with respect to time at a given point is defined by Favre (1969) as

\[
\hat{u}_i = \lim_{T \to \infty} \frac{1}{2T} \int_{-T}^{T} \rho \, u_i \, \frac{d}{dt} \, \rho \, dt
\] (2.7)

The instantaneous value of $u_i$ can be written as

\[
u_i = \hat{u}_i + u''_i
\] (2.8)

where $u''_i$ is the fluctuating component due to the averaging procedure, and

\[
\bar{u''}_i = -\frac{\rho u'_i}{\bar{\rho}} \neq 0
\] (2.9)

\[
\bar{\rho u'_i} = 0
\] (2.10)

Favre averaging has the advantage over Reynolds averaging in that it provides equations describing the mean values of quantities which are conserved. If the density of the flowfield is
constant or the flow is incompressible, then the equations obtained using both averaging tech­
niques are identical. Also, the averaging process should involve ensemble averaging; however, for stationary flows, time averaging and ensemble averaging are identical (Lumly 1970).

Substituting the definition of Favre averaging Eq. (2.8), into the velocity components, and the Reynolds averaging into the static pressure $p$ and the density $\rho$ of the instantaneous flow equations, Eqs. (2.1–2.3), and time averaging leads to the Favre-averaged form of the equations of continuity and conservation of momentum expressed as

mass conservation

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

momentum conservation

$$\frac{\partial}{\partial t} (\bar{\rho} \bar{u}_i) + \frac{\partial}{\partial x_j} (\bar{\rho} \bar{u}_j \bar{u}_i) = - \frac{\partial \bar{\rho}}{\partial x_i} + \frac{\partial \bar{p}}{\partial x_j} - \frac{\partial}{\partial x_j} \rho \bar{u}_i \bar{u}_j''$$

If Reynolds averaging were used, the correlations of density and velocities such as $\rho' \bar{u}'$ and $\rho' u'' v''$ would appear in the above equations, and the resulting governing equations would be more complex. Comparing the set of instantaneous flow equations Eqs. (2.1–2.3) to the Favre-averaged equations Eqs. (2.11 and 2.12), some new terms involving fluctuations appear in the Reynolds-averaged equations which originate from turbulence. Consider the momentum equations which relate the particle acceleration to stress gradients. The new terms $\frac{\partial}{\partial x_j} (\rho \bar{u}_i' \bar{u}_j'')$ are the apparent stress gradients due to turbulent motion since the rest of the acceleration terms have already been identified for the time-mean motion. It is conventional to take this new term to the right-hand side and interpret it as a stress rather than an acceleration term. In most turbulent flows, this term dominates over the viscous counterpart to become the sole mechanism for diffusive momentum transport.

The Favre and time averaged continuity and momentum equations become the governing equations for turbulent flows. However, this set of equations is not closed due to the unknown Reynolds stress tensor $\rho \bar{u}_i' \bar{u}_j''$. One possible approach, which is known as the Reynolds stress closure, is to solve transport equations for each of the Reynolds stress components. The Reynolds stress equations are obtained by multiplying the instantaneous momentum equation Eq. (2.2) by $w_j''$ and adding to it the same equation with suffices $i$ and $j$ interchanged and then time averaging the resulting equation. However, further higher order correlation terms arise and must be solved themselves or modeled. Because of the complexity and the large amount of computational efforts required, the model has not been widely used. Another alternative to the closure problem is to impose certain assumptions to the stress transport equations so that they be reduced to a set of algebraic expressions. This algebraic stress model represents a significant simplification over the Reynolds stress model. However, the equation formulation also entails substantial algebraic manipulation and special care is required when incorporating these stresses into the mean flow equations to achieve stability and convergence.
2.2 Two-Equation Model $k-\varepsilon$

Reynolds stresses can be interpreted as the diffusion fluxes of momentum due to turbulence, hence they can be expressed in terms of gradient transport hypothesis which are analogous to the stress-strain tensor for laminar flow. The main advantage of this approach is that it is low in computing cost and time for no equations need to be solved for the Reynolds stresses themselves. This approach is called the mean flow closure and it is achieved by relating the Reynolds stresses to the mean strain rate through the Boussinesq approximation (1877)

$$
-\rho u_i^\prime u_j^\prime = \mu_t \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \left( \mu_t \frac{\partial \bar{u}_t}{\partial x_t} + \bar{p} \right) \delta_{ij}
$$

(2.13)

where $\mu_t$ is the turbulent viscosity and $k$ is the density-weighted turbulent kinetic energy defined as

$$
k = \frac{\rho u_i^\prime u_i^\prime}{2\bar{p}}
$$

(2.14)

Owing to the definition of $k$ in Eq. (2.14), an extra term of $-\frac{2}{3} \bar{p}k \delta_{ij}$ has to be inserted to the right hand side of Eq. 2.13 to have both sides balanced. Hence, this approximation of the Reynolds stresses is not a pure strain rate. By analogy with kinetic theory of gases for molecular viscosity, the turbulent viscosity can be modeled as

$$
\mu_t = \rho V_t \ell
$$

(2.15)

where $V_t$ and $\ell$ are the characteristic velocity and length scale of turbulence, respectively. Many different theories have been developed for these parameters (Launeler and Spalding 1972). One of the most frequently used is the $k-\varepsilon$ model first proposed by Harlow and Nakayama (1968), and developed by Jones and Launder (1972).

In the mixing-length hypothesis, Prandtl (1925) proposed that the characteristic velocity of turbulence be

$$
V_t = \ell \left| \frac{\partial u}{\partial y} \right|
$$

(2.16)

This implies that the turbulent viscosity vanishes whenever the mean velocity gradient is zero. Because of this shortcoming of the mixing length hypothesis, an alternative formulation of the characteristic turbulence velocity is necessary. Prandtl (1945) and Kolmogorov (1942) proposed that the random velocity of turbulence should relate to a turbulence property itself instead of a mean velocity gradient. They chose the characteristic turbulence velocity as the square root of the time-averaged turbulence kinetic energy

$$
V_t \propto \sqrt{k}
$$

(2.17)

where $k$ is determined from the solution of a transport equation. Although a transport equation can also be developed for the length scale of turbulence, the terms contained in such an equation would not be easily modeled. Some researchers had more success by solving a transport equation for a length scale related parameter instead of the length scale itself (Launeler
and Spalding 1972). Chou (1945), Davidov (1961), Harlow-Nakayama (1968) and Jones and Launder (1972) have favored the turbulence kinetic energy dissipation rate $\varepsilon$ as the second variable because of the relative ease with which the exact equations for $\varepsilon$ can be derived and that $\varepsilon$ itself appears directly as an unknown in the $k$ transport equation. Jones and Launder assumed that the inviscid turbulence kinetic energy dissipation rate is related to other parameters through

$$\varepsilon \sim \frac{k \cdot k^{1/2}}{\ell}$$

(2.18)

Hence, the length scale of turbulence is defined by this model as

$$\ell = C_{\mu} \frac{k^{3/2}}{\varepsilon}$$

(2.19)

where $C_{\mu}$ is a constant of proportionality and the turbulence viscosity is expressed as

$$\mu_t = C_{\mu} \rho \frac{k^2}{\varepsilon}$$

(2.20)

The effective viscosity is then defined as

$$\mu_{\text{eff}} = \mu + \mu_t$$

(2.21)

The $k-\varepsilon$ model is the simplest model which is suitable for calculations of turbulent recirculating flows. Its success lies in the allowance of the length scale of turbulence of a range of complex flowfield to be determined and not prescribed. Considerable experience has been gathered by many workers with this model, such as free shear flows by Launder, et al. (1973), recirculating flows by Sindir (1982) and Lilley (1985), and confined reacting flows with and without swirl by Smith and Smoot (1981), and Nikjoo and So (1987).

To obtain the turbulence kinetic energy transport equation, multiply the instantaneous momentum equations by $u_i$, sum the resulting equations and then apply Favre averaging. The standard model form of the $k$-transport equation is

$$\frac{\partial}{\partial t}(\bar{p}k) + \frac{\partial}{\partial x_i} \left( \bar{p}u_i k - \frac{\mu_{\text{eff}}}{\sigma_k} \frac{\partial k}{\partial x_i} \right) =$$

$$\mu_t \frac{\partial \bar{u}_i}{\partial x_j} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \frac{\partial \bar{u}_\ell}{\partial x\ell} \left( \mu_t \frac{\partial \bar{u}_\ell}{\partial x\ell} + \bar{p}k \right) - \bar{p} \varepsilon$$

(2.22)

where $\sigma_k$, an empirical constant, is the turbulent Prandtl number. The first two terms on the right hand side are commonly known together as the turbulence generation or production term.

The dissipation rate of turbulence kinetic energy transport equation is formed by multiplying the divergence of the $u_i$ transport equation by $2\mu \frac{\partial u_i}{\partial x_j}$ and time averaging. Following the modeling of the second and third correlation terms as suggested by Launder, Reece and
Rodi (1975), three empirical constants are introduced and the standard $\varepsilon$-transport equation becomes

$$
\frac{\partial}{\partial t}(\bar{\rho}\varepsilon) + \frac{\partial}{\partial x_i}\left(\bar{\rho}\bar{u}_i\varepsilon - \frac{\mu_{eff}}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i}\right) = 
$$

$$
\frac{\varepsilon}{k} \left\{ C_1 \left[ \mu_t \frac{\partial \bar{u}_i}{\partial x_j} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \frac{\partial \bar{u}_l}{\partial x_l} \left( \mu_t \frac{\partial \bar{u}_l}{\partial x_l} + \bar{p}k \right) \right] - C_2 \bar{p}\varepsilon \right\} \quad (2.23)
$$

where $C_1$, $C_2$ and $\sigma_\varepsilon$ are empirical constants with

$$
\sigma_\varepsilon = \frac{\kappa^2}{(C_2 - C_1) C_\mu^{1/2}} \quad (2.24)
$$

where $\kappa$ is the von Karman constant. Constants employed in the $k-\varepsilon$ model are given in Table 1.

Attempts to modify the $k-\varepsilon$ transport equations have been done by some researchers and are described in the remainder of this section. However, only the standard $k-\varepsilon$ model is used in this work. Although the $k-\varepsilon$ model has been successfully applied to many flow problems, it has a few drawbacks. The $k-\varepsilon$ is based on the isotropic effective eddy viscosity concept. In the calculations of complex turbulent flows where the shear stress and the velocity gradients may have different signs, the simple isotropic effective eddy viscosity assumption breaks down, and the effects of the non-equal normal stresses and secondary strain rates may become substantial. The standard $k-\varepsilon$ has been modified by some researchers to account for the effects of low Reynolds number flows, wall layers and streamline curvature. The extension of the $k-\varepsilon$ model to low Reynolds numbers was reported by Jones and Launder (1973) so that the turbulence model equations would be valid for the laminar, transition and fully turbulent flows. To avoid the need for detailed calculations in the near wall regions, Launder and Spalding (1974) introduced wall functions, Chieng and Launder (1980) introduced a detailed two-layer near wall model, and Amano (1984) developed a two-layer and a three-layer near wall model.

These near-wall models of Chieng and Launder, and Amano are useful for improving the prediction of wall heat transfer rates; however they fail to improve the predictions of the flowfield significantly. Iacovides and Launder (1984) proposed the parabolic sublayer (PSL) approach and eliminated the use of wall functions. The successful application of the PSL approach is not only as economical as the wall functions but also produces more realistic results. Recently, Nagano and Hishida (1987) proposed a model function which accounts for low Reynolds number flows and wall-proximity effects, and additional terms which improve predictions in the wall region.

When significant streamline curvatures exist in the flow domain, the $k-\varepsilon$ model cannot take into account the enhanced turbulence diffusion due to the extra strain rates with streamline curvature on turbulence, and methods of incorporating it into the turbulence models were given by Bradshaw (1973). Launder, Priddin and Sharma (1977) introduced corrections to the turbulence length scale, determining the $\varepsilon$-equation to account for curvature effects,
which depend on the gradient of the Richardson number. To account for streamline curvature due to swirl, Launder, Pridin and Sharma (1977), and Srinivasan and Mongia (1980) modified the constant $C_2$ appearing in the dissipation term of the $\varepsilon$–equation by relating them to the Richardson number. Rodi (1979) modified the constant $C_1$, by relating it to the flux Richardson number. The above approaches modify an already modeled term in the $\varepsilon$–equation. Militzer, Nicoll and Alpay (1977) corrected the turbulence kinetic energy production term to take into account the curvature effect. This approach differs from the previous ones in that it actually modifies an exact term in the $k$–equation.

### 2.3 Chemistry Model

Flows with combustion and multiple chemical species involve the complications arising from the turbulence-chemistry interaction phenomena. Since it is costly and time consuming to consider a large number of finite-rate chemical reaction steps and many chemical species, an economical, simple and realistic chemistry model must be implemented in the computer code for the prediction of reactive flows. A combustion model will provide a method of calculating the formation or reaction rate $R_k$ of each chemical species $k$.

Molecular mixing occurs at the interface of the smallest eddies. The chemical species mix at the microscale level because of the large specific surface area. The vortex stretching further contributes to the mixing process and increases the chemical species concentration gradients. For diffusion flames, fuel and oxidant are contained in separate eddies initially. The complete mixing process occurs only after a period equal to the decay time of the eddies. In most engineering problems, the eddy decay time is at least two orders of magnitudes larger than the overall chemical reaction time scale. Hence, typical turbulent combustion processes are mixing controlled instead of chemical kinetics controlled. Several authors have reported success with a small number of chemical reaction steps applied to practical engineering problems, for example Chiappetta (1983), Magnussen and Hjertager (1978), Ramos (1984), Kenworthy, Correa and Burrus (1983), and Hautman et al. (1981).

#### 2.3.1 Conservation of Chemical Species

In a multiple chemical species flow, the mass density of each chemical species $\rho_k$ has to be accounted for to obtain the mass density of the mixture, since

$$\rho = \sum_k \rho_k$$  \hspace{1cm} (2.25)

The conservation equations for each chemical species $k$ is

$$\frac{\partial}{\partial t}(\rho_k) + \frac{\partial}{\partial x_i}(\rho_k u_i + J_k) = R_k$$  \hspace{1cm} (2.26)

Since all flow properties are to be calculated from the corresponding Favre-averaged values, the overbar notation is dropped here. From the law of conservation of mass, there is no generation
or destruction of the total mass due to chemical reactions and

\[ \sum_k R_k = 0 \]  

(2.27)

where \( R_k \) is the formation or reaction rate of chemical species \( k \) due to chemical reactions per unit volume of mixture. The diffusion flux vector \( J_k \) of \( \rho_k \) is defined by the Fick’s law

\[ J_k = - \frac{\mu_t}{\sigma_{Sc_t}} \frac{\partial}{\partial x_i} \left( \frac{\rho_k}{\rho} \right) \]  

(2.28)

Here \( \sigma_{Sc_t} \) is the mixture turbulent Schmidt number, which is a measure of the relative importance of momentum transfer and mass transfer. The net diffusion of all chemical species also obeys the condition of

\[ \sum_k J_k = 0 \]  

(2.29)

The conservation equation of chemical species \( k \) in Eq. (2.26) can be written in terms of mass fraction \( m_k \) as

\[ \frac{\partial}{\partial t}(\rho m_k) + \frac{\partial}{\partial x_i} \left( \rho m_k u_i - \frac{\mu_t}{\sigma_{Sc_t}} \frac{\partial m_k}{\partial x_i} \right) = R_k \]  

(2.30)

since

\[ m_k = \frac{\rho_k}{\rho} \]  

(2.31)

### 2.3.2 One-Step Overall Chemical Reaction

A simple one-step chemical reaction model is chosen. An extension to the two or more step reaction models is straightforward.

The one-step overall and irreversible chemical reaction equation for hydrocarbons is

\[ C_x H_y + (x + \frac{y}{4}) (O_2 + n N_2) \rightarrow x CO_2 + \frac{y}{2} H_2O + (x + \frac{y}{4}) n N_2 \]  

(2.32)

where \( x \) and \( y \) are the chemical composition of the fuel and \( n \) is the fraction of nitrogen in air. The ratios of formation of the different species by chemical reactions can be expressed as

\[ r_1 = \text{mass of } O_2 / \text{mass of fuel} = (x + \frac{y}{4}) \frac{M_{O_2}}{M_{fu}} \]

\[ r_2 = \text{mass of } CO_2 / \text{mass of fuel} = x \frac{M_{CO_2}}{M_{fu}} \]

\[ r_3 = \text{mass of } H_2O / \text{mass of fuel} = \frac{y}{2} \frac{M_{H_2O}}{M_{fu}} \]  

(2.33)

where \( M_k \) is the molecular weight of species \( k \). A transport equation can be written for each species according to Eq. (2.30) with \( k \) and \( R_k \) as shown in Table 2. Because the formation rates of various species are related via the above ratios, the mass fractions of the species can
be added in certain proportions to yield zero formation rate. Thus a new function can be
defined that gives no source term in the transport equation. It is called the mixture fraction
\( f \) defined as
\[
f = \frac{m_{\text{CO}_2}}{r_2} + m_{f_u} \tag{2.34}
\]
Since \( R_k \) for the \( \text{O}_2 \), \( \text{CO}_2 \) and \( \text{H}_2\text{O} \) species are proportional to \( R_{f_u} \) only, the transport equations
for \( m_{f_u} \) and \( f \) need to be solved and the remaining \( m_k \) can be determined from Eq. (2.33).

The composition of the oxidant is taken as 79\% \( \text{N}_2 \) and 21\% \( \text{O}_2 \). The mass fractions of
\( \text{N}_2 \) and \( \text{O}_2 \) in the oxidant are thus 0.767 and 0.233, respectively. If the transport equations
of \( m_{f_u} \) and \( f \) are solved and defined in any part of the flow, then the mass fractions of the
species, besides fuel, are determined by
\[
\begin{align*}
m_{\text{CO}_2} &= (f - m_{f_u}) \left( x \frac{M_{\text{CO}_2}}{M_{f_u}} \right) \\
m_{\text{H}_2\text{O}} &= (f - m_{f_u}) \left( \frac{y}{2} \frac{M_{\text{H}_2\text{O}}}{M_{f_u}} \right) \\
m_{\text{N}_2} &= 0.767 (1 - f) \\
m_{\text{O}_2} &= 1 - (m_{f_u} + m_{\text{CO}_2} + m_{\text{H}_2\text{O}} + m_{\text{N}_2}) \tag{2.35}
\end{align*}
\]
since
\[
\sum_k m_k = 1 \tag{2.36}
\]

### 2.3.3 Chemical Mean Formation Rate Models

Turbulent reactive flows are difficult to predict because neither the turbulent chemical trans­
port nor the turbulent chemical kinetics are well understood. Thus turbulence/chemistry
interaction is generally poorly predicted. Therefore, of primary importance is the evaluation
of the mean formation rates of the chemical species which affect the heat release in the flow
and influence the turbulence structure. Since the chemical kinetic equations are nonlinear
in temperature and concentrations, incorrect mass-averaging of various terms can cause large
errors. The modeling of turbulent combustion in the present work is the mean formation rate
approach which determines the mass fractions of the species directly from transport equations
and the formation rate model for \( R_k \). It is applicable to both diffusion and pre-mixed flames.
The only problem in the mean formation rate approach is the modeling of \( R_k \).

#### The Finite-Rate Chemistry Approach

The instantaneous formation rate of species \( k \) is represented by the Arrhenius law
\[
R_k = A m_k m_j \exp \left[ -E/(RT) \right] \tag{2.37}
\]
where \( A \) is the pre-exponential rate constant, \( E \) is the activation energy, \( R \) is the universal
gas constant and \( T \) is the absolute temperature. To obtain the mean formation rate \( \overline{R_k} \),
it is necessary to express each of the variables by their mean and fluctuating components. Application of Reynolds averaging and appropriate simplifications gives an expression for $\overline{R_k}$ which contains correlations between fluctuations in the various quantities. Neglecting these correlations in the evaluation of the $\overline{R_k}$ can affect the computed reaction rate by an order of magnitude or more. Moreover, for any reasonable combustion scheme, many steps of chemical reaction have to be considered. It is obvious that the closure of turbulent reacting flows through such a scheme is a formidable one. Donaldson (1974) and Borghi (1974) attempted to determine the various correlations directly and had success only in flows where the fluctuations and heat release were small.

**The Fast-Chemistry Approach**

Another approach to the non-premixed combustion system is based on the assumption that the chemical reaction time scale is at least two orders of magnitude faster than the eddy decay time. Hence the chemical reaction rates are completely mixing controlled and are function of the turbulence parameters rather than the chemical kinetics.

The eddy break-up (EBU) model of Spalding (1971, 1977) is the main example of this approach. This model can be expressed as

$$\overline{R_{EBU}} = C \frac{\varepsilon}{k} \rho m_k$$

(2.38)

where $C$ is an empirical constant. Its use is based on the assumption that the mean reaction rate is solely determined by the rate of eddy decay through the process of turbulent mixing and vortex stretching. Such a model can easily be applied to multi-step chemical reaction mechanisms.

Although fuel oxidation in combustors is primarily mixing controlled under most operating environments, the chemical kinetic effects should also be included for the prediction of combustion, emissions, and flame stability. In view of the large number of species and reactions taking place, one solution to include finite-rate chemistry is the use of the global approach.

To ensure that both chemical-kinetic and turbulent mixing processes are represented in the mean formation rate, a hybrid model of the global kinetics and turbulent mixing is employed. This model is shown here for a single-step irreversible reaction as

$$\overline{R} = \min [ \overline{R_{ED}}, \overline{R_{CK}} ]$$

(2.39)

$$\overline{R_{ED}} = C \frac{\varepsilon}{k} \rho \min [ \overline{m_{fu}}, \overline{\bar{m}_{o2}} ]$$

(2.40)

$$\overline{R_{CK}} = A \overline{m_{fu}} \overline{\bar{m}_{o2}} p^2 \exp \left[ -\frac{E}{(RT)} \right]$$

(2.41)

Eq. (2.39) implies that the mean formation rate is determined by the lower of the two rates. Inside the square brackets, the first rate represents the turbulent mixing process while the second rate represents the chemical kinetic process. The rate of turbulent mixing shown in Eq. (2.40) is the eddy-dissipation (ED) model of Magnussen and Hjertager (1978) which assumed that the formation rate is proportional to the concentration of the governing species. In non-premixed flames, fuel and oxygen occur in separate eddies. In regions where $\overline{m_{fu}}$ is low
and $\bar{m}_{O_2}$ is high, fuel will be the reacting species that shows the most marked intermittency. Thus the dissipation of the fuel eddies will govern the rate of combustion and this is modeled by the first quantity inside the square brackets of Eq. (2.40). However, in regions where $\bar{m}_{fu}$ is high and $\bar{m}_{O_2}$ is low, oxygen will be the reacting species that shows the most marked intermittency and hence the dissipation of the oxygen eddies will limit the rate of combustion as represented by the second quantity within the squared brackets, with $r_1$ being the stoichiometric oxygen to fuel mass ratio as defined in Eq. (2.33). The empirical constant $C$ in the eddy-dissipation model is given a value of 4 by Magnussen and Hjertager.

The mean formation rate based on global kinetics (GK) according to Spalding (1978) is shown in Eq. (2.41). The pre-exponential constant, the exponents and the activation energy can be obtained by calibrated and experimental laminar flame speeds.

### 2.4 Conservation of Energy

The energy equation is expressed by

$$\frac{\partial}{\partial t} \left[ \rho \left( H - \frac{\bar{p}}{\rho} \right) \right] + \frac{\partial}{\partial x_i} \left[ \rho u_i H + \bar{q} - \bar{\tau} \cdot u_i \right] = 0 \quad (2.42)$$

where $H$ is the specific stagnation enthalpy of the mixture defined as

$$H = \sum_k m_k h_k + \frac{1}{2} (u_i \cdot u_i) \quad (2.43)$$

with $m_k$ and $h_k$ being the mass fraction and specific enthalpy of species $k$. The stress tensor $\bar{\tau}$ is defined as

$$\bar{\tau} = \tau_{ij} \bar{\dot{e}}^i \bar{\dot{e}}^j \quad (2.44)$$

The quantity included in the square brackets denotes the rate of change of internal energy per unit volume. For a steady flow, the energy equation reduces to the balance of the energy of the mass of gas mixture, the heat transferred to the mass of gas and the shear work performed by surface forces.

In binary or multi-component mixtures, the heat flux vector is given by

$$\bar{q} = -\lambda \nabla T + \rho \sum_k m_k h_k \bar{V}_k \quad (2.45)$$

which is the heat energy due to thermal conduction and thermal diffusion due to temperature gradient. The thermal conductivity $\lambda$ can be expressed in terms of Prandtl number $Pr$ as

$$\lambda = \frac{\mu}{Pr} \sum_k m_k C_{pk} \quad (2.46)$$

where $C_{pk}$ is the specific heat of species $k$. $\bar{V}_k$ is the average velocity of species $k$ relative to the mass-average velocity of the gas mixture. Applying Fick's law

$$\bar{V}_k = -D \nabla (\ln m_k) \quad (2.47)$$
The diffusion coefficient $D$ for all species is the same and introducing the Lewis number

$$Le = \frac{Sc}{Pr}$$

where $Sc$ is the Schmidt number defined as

$$Sc = \frac{\mu}{\rho D}$$

then the heat flux vector can be expressed as

$$\vec{q} = -\frac{\mu}{Pr} \sum_k (m_k C_{pk}) \nabla T - \frac{\mu}{Pr} Le \sum_h h_k \nabla m_k$$

The radiative energy transport term is being neglected in the heat flux term.

In many flows $Le$ is very near unity and is often slightly below unity in combustion gas mixtures, therefore assuming $Le = 1$ is justified in theoretical analyses of combustion. The energy equation can be now written as

$$\frac{\partial}{\partial t} \left[ \rho \left( H - \frac{p}{\rho} \right) \right] + \frac{\partial}{\partial x_i} \left( \rho u_i H - \frac{\mu}{Pr} C_p \nabla T - \frac{\mu}{Pr} \sum_k h_k \nabla m_k - \vec{\tau} \cdot \vec{u}_i \right) = 0 \quad (2.51)$$

where $C_p$ is the mean specific heat expressed as

$$C_p = \sum_k m_k C_{pk}$$

Taking the divergence of Eq. (2.43) gives

$$\frac{\partial}{\partial x_i} \left( H - \frac{u_i^2}{2} \right) = C_p \nabla T + \sum_k h_k \nabla m_k \quad (2.53)$$

The energy equation thus becomes

$$\frac{\partial}{\partial t} \left[ \rho \left( H - \frac{p}{\rho} \right) \right] + \frac{\partial}{\partial x_i} \left\{ (\rho u_i H) - \frac{\partial}{\partial x_i} \left[ \frac{\mu}{Pr} \left( H - \frac{u_i^2}{2} \right) \right] \right\} = \nabla \cdot (\vec{\tau} \cdot \vec{u}_i) \quad (2.54)$$

The mean energy for turbulent combusting flows is derived by density-weighted or Favre averaging (Bilger 1975 and Jones 1979). Applying Reynolds averaging to the density, pressure and stress terms, and Favre averaging to the rest of the flow variables in the energy equation above, results in second-order correlations $\rho u_i'' H''$ in the averaging procedures which are interpreted as diffusion fluxes of energy due to turbulence fluctuations. These correlations can be expressed in terms of $\rho u_i'' u_i''$ and $\rho u_i'' h''$. The velocity correlations are linked to the mean velocity gradient via the Boussinesq approximation as shown in Eq. (2.13). By analogy to the turbulent diffusion fluxes, the second-order correlations between velocity and scalar functions can be represented in terms of the gradients of mean values. Hence,

$$-\rho u_i'' h'' = \frac{\mu_e}{Pr_e} \frac{\partial h}{\partial x_i}$$

(2.55)
Taking the average of the resulting form yields the mean equation for specific stagnation energy
\[ \frac{\partial}{\partial t} \left[ \bar{p} \left( \bar{H} - \frac{p}{\rho} \right) \right] + \frac{\partial}{\partial x_i} \left\{ (\bar{p} \bar{u}_i \bar{H}) - \frac{\partial}{\partial x_i} \left[ \Gamma_H \left( \bar{H} - \frac{\bar{u}_i^2}{2} \right) \right] \right\} = \nabla \cdot (\tau - \bar{\tau}_i) - \nabla \cdot (\Gamma_H \nabla k) \] (2.56)
where the specific stagnation energy of mixture \( \bar{H} \) is defined as
\[ \bar{H} = \sum_k \bar{m}_k \bar{h}_k + \frac{1}{2} (\bar{u}_i \cdot \bar{u}_i) + k \] (2.57)
and \( \Gamma_H \) is the effective exchange coefficient for heat transfer defined as
\[ \Gamma_H = \frac{\mu}{Pr} + \frac{\mu_t}{Pr_t} \] (2.58)

### 2.5 Thermophysical Properties

The laminar viscosity is evaluated by the interpolation formula given by van Driest (1952) based on the Sutherland's theory of viscosity which is expressed as
\[ \frac{\mu}{\mu_0} = \left( \frac{T}{T_0} \right)^{3/2} \frac{T_0 + S_1}{T + S_1} \] (2.59)
where \( \mu_0 \) denotes the viscosity at the reference temperature \( T_0 \), and \( S_1 \) is an empirical constant which assumes the value \( S_1 = 110 \) for air. The laminar Prandtl number is assumed to be constant. The mean density is evaluated from the equation of state for an ideal gas mixture given by
\[ \rho = \frac{p}{RT \sum_k \frac{\bar{m}_k}{M_k}} \] (2.60)
\( R \) being the universal gas constant. The specific enthalpy for each species \( k \) is
\[ h_k = h_k^0 + \int_{T_{ref}}^{T} C_{pk} dT \] (2.61)
Gordon and McBride (1971) curve fitted the specific enthalpy data of each species by a thermodynamic function of temperature with fifth degree polynomial
\[ \frac{h_k}{R} = a_1 T + a_2 \frac{T^2}{2} + a_3 \frac{T^3}{3} + a_4 \frac{T^4}{4} + a_5 \frac{T^5}{5} + a_6 \] (2.62)
Two sets of coefficients for \( a_1 \) to \( a_6 \) are given for the temperature intervals of 300 to 1000 K and 1000 to 5000 K with the constraint that the data is equal at 1000 K.

To determine the temperature \( T \), an interactive scheme is used. First, the mean mixture specific enthalpy \( \bar{h} \) is calculated from Eq. (2.57) which is defined as
\[ \bar{h} = \sum_k \bar{m}_k \bar{h}_k \] (2.63)
Since \( \hat{n}_k \) of all species are obtained from the chemical transport equations and algebraic relations, a guess of the temperature will give all the \( h_k \) and subsequently an estimated mean mixture specific enthalpy. An iterative algorithm is then used to obtain a temperature which satisfies the given value of \( \hat{h} \) from Eq. (2.57).

### 2.6 General Differential Equation

All the governing transport equations introduced in the foregoing sections indicated that all the dependent variables obey a generalized conservation principle. Since each dependent variable \( \phi \) has been formulated to represent an average value, and the resulting transport equations are identical, the overbar notations can be dropped. The differential equations are all of the form

\[
\frac{\partial}{\partial t} (\rho \phi) + \nabla \cdot (\rho u_t \phi - \Gamma_\phi \nabla \phi) = S_\phi
\]

where \( \phi \) is the dependent variable, \( \Gamma_\phi \) is the diffusion coefficient or the effective exchange coefficient, and \( S_\phi \) is the source term.

To formulate any particular differential equation into the general differential form of Eq. (2.64), manipulate the original differential equation until the unsteady, convection and diffusion terms conform to the standard form. The coefficient of \( \nabla \phi \) in the diffusion term is then considered to be the expression for \( \Gamma_\phi \). All of the remaining terms are then cast into the source term \( S_\phi \) and put on the right-hand side of the general equation. The variables \( \phi \) can be of any dimensional quantity, as well as any dimensionless variable.

The four terms in the general differential equation Eq. (2.64) are the unsteady or transient term, the convection term, the diffusion term and the source term. The dependent variable \( \phi \) can represent many different quantities such as mass, a velocity component, the turbulence kinetic energy, the turbulence dissipation rate, the mass fraction of a chemical species, or the enthalpy of the flow. For each of these dependent variables, \( \Gamma_\phi \) and \( S_\phi \) take on different forms and meanings. The various \( \phi \) and their corresponding \( \Gamma_\phi \) and \( S_\phi \) for a 2-D formulation are tabulated in Table 3. The governing transport differential equations in Cartesian coordinates for two-dimensional steady state flows are given in Appendix A.

By casting all governing differential equations into the general form, there is a need to search for the numerical solution of only one differential equation (the standard form). It is also sufficient to construct a computer program which can be repeatedly used to solve various \( \phi \) with the corresponding \( \Gamma_\phi \), \( S_\phi \) and the appropriate initial and boundary conditions. Hence, the concept of a general differential equation enables to save time and effort in arriving at a general numerical algorithm and in preparing a general computer program.
2.7 Wall Functions

The \( k-\varepsilon \) turbulence model is devised for high Reynolds number flows where the turbulence structure is assumed to be independent of laminar viscosity. Near a solid boundary where the velocities become smaller, the influence of the laminar viscosity is significant and the variation of turbulence and flow properties becomes appreciable. Hence, near the vicinity of solid boundaries, the previously neglected low Reynolds number effects become prominent and should be taken into account. This can be achieved either by solving the low Reynolds number form of the transport equations which requires the use of very fine computational grids, large computer storage space and vast amounts of computer time due to the inherent slow convergence characteristics of the low Reynolds number models as described by Chieng and Launder (1980), or by developing wall functions that modify the high Reynolds number models in an indirect fashion through the specification of boundary conditions.

The approach employed in the TEACH code is a two-layer near-wall model which links the flow conditions in the interior fully turbulent region to the flow variables at the wall. The wall functions are derived from the steady one-dimensional Couette flow analysis of the wall region.

2.7.1 Near Wall Velocity Profile

According to Hinze (1959), the wall region can be divided into three zones from the analysis of the experimental data: a viscous sublayer (\( 0 < Y^+ < 5 \)) adjacent to the wall with substantial viscous effects; a buffer layer (\( 5 < Y^+ < 30 \)) where the flow is neither completely turbulent nor entirely dominated by viscous effects; and an inertial sublayer (\( 30 < Y^+ < 400 \)) where the flow is assumed to be entirely turbulent but the shear stress \( \tau \) is assumed equal to the wall shear stress \( \tau_w \). Thus the flow conditions across the wall layer can be given as functions of the local Reynolds number \( Y^+ \) as defined by Launder and Spalding (1974) as

\[
Y^+ = \frac{\rho u_r Y}{\mu}
\]

where \( Y \) is the normal distance from the wall boundary and \( u_r \) is the friction velocity defined with the wall shear stress \( \tau_w \) as

\[
u_r = \sqrt{\frac{\tau_w}{\rho}}
\]

The common engineering practice is to extend both the viscous and inertial sublayers to a mutual boundary at \( Y^+ = 11.63 \) and to dispense with the buffer layer (Gosman and Pun 1974). Hence beyond this mutual boundary, the flow is assumed to be completely turbulent and the velocity profile is described by the logarithmic law of the wall (Schlichting 1960), and within the viscous sublayer, the flow is assumed to be completely viscous with the velocity profile being linear. Therefore the velocity profile \( U \) across the wall layer is assumed as follows:

\[
\frac{U}{u_r} = \begin{cases} Y^+ & Y^+ \leq 11.63 \\ \frac{\ln (E Y^+)}{\kappa} & Y^+ > 11.63 \end{cases}
\]
where $E$ is an integration constant that depends on the magnitude of the variation of the shear stress across the wall layer and on the roughness of the wall. Given in Table 1 are the values of $E$ and $\kappa$ for smooth and impermeable walls with constant shear stresses from experiments.

### 2.7.2 Near Wall Shear Stress

The algebraic expression for the wall shear stress within the fully turbulent sublayer can be derived with the assumption that turbulence is in local equilibrium in the near wall region of a Couette flow (Launder and Spalding 1972). In the fully turbulent sublayer ($Y^+ > 11.63$), the shear stress is approximated by the wall shear stress

$$\tau \approx \tau_w = \mu_t \frac{\partial U}{\partial Y}$$

(2.68)

as $\mu_t$ dominates and $\mu$ is assumed negligible. Since the convection and diffusion of $k$ are always negligible in the near wall region, the transport equation for $k$ reduces to a balance between the local production and dissipation of turbulence kinetic energy, resulting in

$$\mu_t \left( \frac{\partial U}{\partial Y} \right)^2 = \rho \varepsilon$$

(2.69)

Replacing the velocity gradient and the dissipation rate terms with Eqs. (2.18) and (2.68) in the above equation, the wall shear stress can be written as

$$\tau_w = C_{\mu}^{1/2} \rho k$$

(2.70)

From Eqs. (2.66), (2.67) and (2.70),

$$\tau_w = -\kappa C_{\mu}^{1/4} \rho k^{1/2} U \frac{1}{\ln (EY^+)}$$

(2.71)

where

$$Y^+ = \frac{C_{\mu}^{1/4} \rho k^{1/2} Y}{\mu}$$

(2.72)

A negative sign is inserted in Eq. (2.71) since $\tau_w$ and $U$ must have opposite directions.

In the viscous sublayer ($Y^+ \leq 11.63$), the wall shear stress is assumed to have the profile of

$$\tau_w = -\frac{\mu U}{Y}$$

(2.73)

with turbulent viscosity assumed negligible.

The total tangential velocity $U$ is the resultant of the two tangential velocity components near the wall. For a wall in the $x$-$z$ plane,

$$U = \sqrt{u^2 + w^2}$$

(2.74)
The components of the shear stress are then obtained by resolving the resultant shear stress in the appropriate coordinate directions. For example, the shear stress $\tau_{xy}$ and $\tau_{zy}$ in the directions of $x$ and $z$ respectively at a wall normal to the $y$-direction are

$$\tau_{xy} = \tau_w \frac{u}{U} \quad \text{and} \quad \tau_{zy} = \tau_w \frac{w}{U} \quad (2.75)$$

### 2.7.3 Wall Functions for the Reynolds Equation

In the near wall region, the flow velocity adjacent to the solid wall is not given directly from the results obtained in Eq. (2.67). The near wall velocities are still obtained from the solution of the Reynolds Eq. (2.12). Although there is no convective flux through the cell boundary that faces the wall, the diffusion term in the Reynolds equation is approximated by the wall shear stress through the wall function.

For example, the north wall diffusion flux of the $x$-momentum equation is $\mu_{\text{eff}} \frac{\partial \bar{u}}{\partial y}$. The $\tau_{xy}$ component of $\tau_w$ is given by

$$\tau_{xy} = \mu_{\text{eff}} \left( \frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right) \quad (2.76)$$

However, $\frac{\partial \bar{v}}{\partial x}$ can be neglected since it approaches zero near the north wall. The diffusion flux term needed only to be concerned with is $\tau_{xy}$. Thus the wall function for the $x$-momentum equation adjacent to the north wall is

$$\mu_{\text{eff}} \frac{\partial \bar{u}}{\partial y} = \tau_{xy} = \begin{cases} -\frac{\bar{u}}{Y} & Y^+ \leq 11.63 \\ -\kappa C_{\mu}^{1/4} \rho_0 \frac{1}{\rho} k^{1/2} \bar{u} & \ln (EY^+) \\ \ln (EY^+) & Y^+ > 11.63 \end{cases} \quad (2.77)$$

Similar expressions are obtained for other momentum diffusion terms for other near wall cell faces.

### 2.7.4 Wall Functions for $k$ and $\varepsilon$

The near wall turbulence kinetic energy is obtained from the solution of the $k$ transport equation Eq. (2.22). As discussed in paragraph 2.7.2, in the near wall region, the $k$ transport equation reduces to a balance between the local production and dissipation of $k$. From Eqs. (2.20, 2.68 and 2.69),

$$\frac{\rho \frac{k}{\tau_w}}{C_{\mu}^{-1/2}} = \text{constant} \quad (2.78)$$

near wall which implies that the wall flux is zero. However, the production and dissipation terms need to be modified to include the wall effects by substituting the shear stress terms expressed in velocity gradients with the wall functions given by Eq. (2.77).
For the derivation of the near wall turbulence dissipation rate, the length scale is assumed to be proportional to the normal distance from the wall. The turbulence viscosity term in Eq. (2.69) is substituted by Eq. (2.20) resulting in

\[
\varepsilon^2 = C_\mu k^2 \left( \frac{\partial U}{\partial Y} \right)^2
\]  

Then differentiate \( U \) with respect to \( Y \) in Eq. (2.67) to get

\[
\frac{\partial U}{\partial Y} = \frac{u_\tau}{\kappa Y}
\]

Substituting into Eq. (2.79)

\[
\varepsilon^2 = C_\mu k^2 \left( \frac{u_\tau}{\kappa Y} \right)^2
\]  

Using Eqs. (2.66), (2.70) and (2.81),

\[
\varepsilon = \frac{C_\mu^{3/4} k^{3/2}}{\kappa Y}
\]

This is the effective boundary condition for \( \varepsilon \) to be prescribed at the near wall points.

The \( p\varepsilon \) source term in the \( k \) and \( \varepsilon \) transport equations when integrated over a control volume \( V \) extending up to the wall takes the form

\[
\int_V p\varepsilon \, dV = \int_V \tau_w \frac{\partial U}{\partial Y} \, dV
\]

\[
\approx \frac{\tau_w U V}{Y}
\]

\[
= \rho C_\mu^{3/4} k^{3/2} \left( \frac{U}{u_\tau} \right) \frac{V}{Y}
\]

using the relations of Eqs. (2.66–2.72).

### 2.7.5 Wall Function for \( \tilde{H} \)

The heat flux in the near wall region is assumed to be constant as Gosman and Ideriah (1976) suggested

\[
Q \approx Q_w
\]  

The heat flux across the wall layer can be derived from the Couette flow. The one dimensional energy equation in the absence of radiation, pressure gradients and surface roughness is written as

\[
\frac{d}{dy} (\bar{\rho} \bar{v} \bar{H}) + \frac{d}{dy} \left( J_H - u_\tau \right) = 0
\]  

From the continuity equation,

\[
\bar{\rho} \bar{v} = \text{constant} = \bar{\rho}_w \bar{v}_w
\]
with the subscript \( w \) representing the wall value. Hence the energy equation becomes
\[
\tilde{\rho}_w \tilde{v}_w (\tilde{H} - \tilde{H}_w) = -(J_H - u_r - J_{Hw})
\]
\[
- J_H = - J_{Hw} + \tilde{\rho}_w \tilde{v}_w (\tilde{H} - \tilde{H}_w) - \mu \cdot \mu_{\text{eff}} \frac{d \tilde{u}}{d y}
\] (2.87)

From the transport law of energy,
\[
- J_H = \Gamma_{H\text{eff}} \left[ \frac{d \tilde{H}}{d y} - \frac{d (\tilde{u}^2/2)}{d y} \right]
\] (2.88)

Therefore
\[
\frac{d \tilde{H}}{d y} = - J_{Hw} + \tilde{\rho}_w \tilde{v}_w (\tilde{H} - \tilde{H}_w) + \frac{1 - \sigma_{H\text{eff}}}{\Gamma_{H\text{eff}}} \frac{d (\tilde{u}^2)}{d y}
\] (2.89)

where \( \Gamma_{H\text{eff}} \) and \( \sigma_{H\text{eff}} \) are the effective exchange for heat transfer and effective mixture Prandtl number respectively. For solid non-permeable walls, there is no mass transfer at the wall, then
\[
\tilde{\rho}_w \tilde{v}_w = 0
\] (2.90)

and Eq. (2.89) becomes
\[
\frac{d \tilde{H}}{d y} = - \frac{J_H}{\Gamma_{H\text{eff}}} + \frac{1 - \sigma_{H\text{eff}}}{2} \frac{d (\tilde{u}^2)}{d y}
\] (2.91)

If the wall temperature is given (\( \tilde{H}_w \) is known), then integrating Eq. (2.91) will yield the wall heat flux
\[
J_{Hw} = \begin{cases} 
\frac{\mu}{\text{Pr}} \frac{(\tilde{H}_w - \tilde{H})}{Y} + (1 - \text{Pr}) \frac{\tilde{u}^2}{2} & Y^+ \leq 11.63 \\
\frac{\rho C_{\mu}^{1/4} k^{1/2}}{\text{Pr}_t} (\tilde{H}_w - \tilde{H}) \left[ \frac{\ln (E Y^+)}{\kappa} + P \right]^{-1} & Y^+ > 11.63 \\
+ (1 - \text{Pr}_t) \frac{\tilde{u}^2}{2} 
\end{cases}
\] (2.92)

For \( Y^+ > 11.63 \), the heat flux is the inverse of a logarithmic function of \( Y^+ \) and the integration constant expressed as a \( P \)-function. There are many forms of the \( P \)-function. Jayatillaka (1969) suggested a \( P \)-function for impermeable, smooth walls without kinetic heating and other heat sources as
\[
P = 3.68 \frac{\text{Pr}}{\kappa} \left( \frac{\text{Pr}}{\text{Pr}_t} - 1 \right) \left( \frac{\text{Pr}_t}{\text{Pr}} \right)^{1/4}
\] (2.93)

In case of an adiabatic wall, the wall heat flux is zero.
2.8 Boundary Conditions

The mathematical description of any particular physical problem in a complete form requires the specification of the conditions at the boundaries and the initial conditions of the physical domain. Although the initial conditions are critical for unsteady flow problems only, the boundary conditions need to be specified at all the walls, inlets and outlet planes, and axes of symmetry in the case of axisymmetric flow.

Inlet Plane

The values of \( u_i, k, p, T \) and \( m_k \) are specified at the inlets. Since it is difficult to obtain measurements of \( \varepsilon \) from experiments, one approach is to specify \( \varepsilon \) via \( k \) and turbulence length scale \( \ell \) as

\[
\varepsilon = \frac{C^{3/4} k^{3/2}}{\ell} \tag{2.94}
\]

where \( \ell \) is assumed to be a small fraction of the inlet dimension. The present approach is to assume that the Reynolds number at the inlets is

\[
Re = 1000 \tag{2.95}
\]

and \( \varepsilon \) can then be calculated from the definition of \( Re \) and Eq. (2.20). Density \( \rho \) is determined from the equation of state in Eq. (2.60). The specific stagnation energy \( \hat{H} \) is evaluated from Eq. (2.57).

Axes of Symmetry

In the case of axisymmetric flow, zero normal gradient condition is specified for all variables except the normal velocity which is zero itself.

Walls

All solid walls are assumed as impermeable and no-slip surfaces. Velocities and other variables may be set to zero except \( \rho, \mu, T \) and \( \hat{H} \) which are given with their appropriate values. The wall functions based on the logarithmic-law of the wall is invoked to determine the wall shear stresses and the values of \( k \) and \( \varepsilon \) close to the wall. Because the velocities are specified at any wall boundary as zero, no specification of \( \rho \) is required. If \( T \) is given at the wall, \( \hat{H}_w \) can be determined and wall heat flux can be calculated from the wall function. However, if the wall is considered adiabatic, a zero wall heat flux is specified.

Outlet Plane

Normal gradient conditions are imposed for all variables for the normal velocity where a special treatment is applied to ensure overall mass conservation.
Chapter 3

Numerical Technique

The set of governing equations, the turbulence and combustion models, wall functions and boundary conditions which describe the complete physical problem in mathematical terms require a numerical procedure to solve the time-dependent or the steady-state behavior of the turbulent and/or reactive flows. A TEACH-type STARPI C computer code (Lilley and Rhode 1982) has been extensively modified in order to solve the complex flowfield of a practical gas turbine combustor under steady state. The resulting two-dimensional code is capable of solving very complex combustor geometry using stair-steps which includes top (north), bottom (south), forward-facing (east) and backward-facing (west) wall boundaries and inlet planes. The calculations of the pressure field through the SIMPLE procedure has been replaced by the faster converging SIMPLEC procedure. The energy equation and the combustion model have also been implemented into the computer program.

The linkages amongst all the transport equations provide a high degree of non-linearity to the problem. The solution procedure must then take into account the interactions amongst all the variables and ensure that successive adjustments of every variable will lead to convergence of the solutions.

3.1 Staggered Grid System

The partial differential equations are discretized and the subsequent finite difference equations are solved on a complex mesh which is composed of three sets of grids staggered together as shown in Fig. 1. The grid lines are drawn in after the boundary has been represented by stair steps. The stair steps will be located in the middle of two grid lines. The physical domain is then discretized into control volumes or cells of finite size formed by locating the cell boundaries midway between grid lines. The intersections of the grid lines, denoted as point $P$, mark the grid nodes where all the variables, except the $u$ and $v$ velocity components, are stored. The $u$ and $v$ velocity components are stored at midway between grid intersections, which coincide with the control volume faces. This positioning has the advantage that the velocity components are directly available for calculation of the convection term through the
walls of the scalar cell, and the $u$ and $v$ momentum control volumes are placed in such a way that the boundaries normal to the velocity direction coincide with the pressure nodes.

Hence there are three sets of grids and three sets of control volumes. The $c$ cells, as shown in Fig. 1(a), for scalars and circumferential velocity component $w$ in swirling flows are defined such that the east and west faces of control volumes are coincident with the longitudinal grid line locations of the longitudinal velocity $u$, and the north and south faces are coincident with the transverse grid line locations of the transverse velocity $v$. The $u$ cells, as shown in Fig. 1(b) are shifted west relative to the $c$ cell such that the east and west faces are coincident with the longitudinal locations of the $c$ grid node. The $v$ cells is similarly shifted south relative to the $c$ cell such that the north and south faces are coincident with the transverse locations of the $c$ grid node as illustrated in Fig. 1(c). Values of variables at cell boundaries are then obtained by averaging the neighboring nodal values if they are not defined at the cell boundaries.

The approach here with control-volume faces located midway between grid points has the consequence that the grid point $P$ does not lie at the geometric center of the control volume. This is a disadvantage because $\phi_P$ is now not considered as a good representative value for the control volume in the calculations of the conductive, diffusive and source terms. Moreover, in the calculation of flux contribution at the control-volume faces, the point $w$ in Fig. 1(a), for example, is not located at the center of the control-volume face. Hence the assumption that the flux at $w$ prevails over the entire control-volume face involves inaccuracy. However, this approach does provide greater accuracy in the evaluations of flux across the faces due to the fact that the slopes of a piecewise-linear profile is identical to the slope of any parabolic profile evaluated midway between two grid nodes. The disadvantage and inaccuracy will be eliminated if the grid spacing in each coordinate direction is equal.

The advantage of using staggered grids for the velocity components are that only realistic velocity fields are acceptable to the continuity equation, and that the pressure lies at the control-volume faces to become the natural driving force for the velocity components. Consequently, physically-realistic solutions will be obtained. However, the employment of staggered grids requires more storage and performs more interpolations of variables between grid nodes. But the benefits it entails justify the additional effort and cost.

3.2 Finite Volume Equation

The general transport equation given by Eq. (2.64) is discretized by the control volume formulation as described by Patankar (1980). The finite volume form for all variables are derived by integrating the parent differential equation over the appropriate control volume with the assumption that the distributions of the variable between nodes care entered in each control volume. The integration of the general differential equation in Eq. (2.64) over a finite control volume $V$ is expressed as

$$\frac{d}{dt} \int_V \rho \phi \, dV + \oint_A (\rho u_i \phi - \Gamma_\phi \nabla \phi) \cdot n \, dA = \int_V S_\phi \, dV \quad (3.1)$$

Considering only the steady state solution, employing the mean-value theorem and lin-
earizing the source term in the vicinity of the center of the control volume, the finite volume equation becomes

$$\left( \rho u \phi - \Gamma_\phi \frac{\partial \phi}{\partial x} \right)_e A_e - \left( \rho u \phi - \Gamma_\phi \frac{\partial \phi}{\partial x} \right)_w A_w + \left( \rho v \phi - \Gamma_\phi \frac{\partial \phi}{\partial y} \right)_n A_n - \left( \rho v \phi - \Gamma_\phi \frac{\partial \phi}{\partial y} \right)_s A_s = (S_P \phi_P + S_U) V \quad (3.2)$$

where subscripts $n, s, e, w$ refer to north, south, east and west cell faces. To aid the convergence of the interactive solution procedure and avoid numerical instability, the diagonal of each equation set is made dominant by casting all the negative source terms in the $S_P$ expression and all the positive source terms in the $S_U$ expression. The source term $S_\phi$ is incorporated in a linearized form with the assumption that it is constant inside the corresponding control volume. The diffusion terms are represented by central differences whereas the convection terms can be represented by various schemes to define fluxes at the faces of the control volume. The hybrid differencing scheme by Spalding (1972) is used in the present code. In this scheme, a Peclet number $Pe$ which is a measure of the relative importance of convection and diffusion is defined, for example at the east face, as

$$Pe_e = \left( \frac{\rho u \delta x}{\Gamma_\phi} \right)_e \quad (3.3)$$

and is calculated at each cell boundary. Central difference is employed for low Peclet number $|Pe| < 2$ and upwind difference is used for high Peclet number $|Pe| \geq 2$. The main advantages of the hybrid scheme are: to give correct limiting values as $Pe$ becomes large or small, to enhance numerical stability of the solution, and to reduce computational cost. According to one-dimensional analysis, the departure of the hybrid upwind solution from the exact (exponential) solution of the convection-diffusion equation is the largest at $|Pe| = 2$. Moreover, the neglect of the diffusion effects as $|Pe|$ exceeds 2 is purely arbitrary.

The general form of the finite volume equation can be written in the final compact form

$$a_P \phi_P = a_n \phi_n + a_s \phi_s + a_e \phi_e + a_w \phi_w + S_U \quad (3.4)$$

where the $a's$ are the total convective and diffusive flux coefficients which are all positive, and

$$a_P = a_n + a_s + a_e + a_w - S_P \quad (3.5)$$

thus linking each value of $\phi$ at node point $P$ with its four neighboring values. The resulting coefficient matrix is thus diagonally dominant. More details on the formulation of the finite volume equation and the hybrid scheme are found in Patankar (1980), Spalding (1972), Lilley and Rhode (1982) and Gosman and Ideriah (1983).

### 3.3 Treatment of Pressure-Linked Equations

The real difficulty in the calculation of the velocity field is due to the unknown pressure field. Even though the pressure gradient term appears in the momentum equation as a source, no
explicit equation exists for the determination of pressure. Hence there is a need to convert the indirect information of the pressure field in the continuity equation into a direct algorithm for the calculation of pressure.

The $u$ momentum equation for the control volume centered at the face of the scalar cell written in the finite-difference equation form, and explicitly showing the pressure term is

$$a_e u_e = \sum a_{nb} u_{nb} + b_e + (p_P - p_E) A_e$$  \hspace{1cm} (3.6)$$

where $p$ is pressure, $A$ is area, $b$ is the positive linearized source term and subscript $nb$ represents the neighboring grid points. Unless the correct pressure field is found, the resulting velocity field will not satisfy the continuity equation. The imperfect velocity field based on a guessed pressure field $p^*$ will be denoted by $u^*$ and satisfy the equation

$$a_e u_e^* = \sum a_{nb} u_{nb}^* + b_e + (p_P^* - p_E) A_e$$  \hspace{1cm} (3.7)$$

The task required is to find a way to improve $p^*$ such that the resulting $u^*$ will progressively satisfying the continuity equation.

Correction of the guessed pressure is then proposed as

$$p = p^* + p'$$  \hspace{1cm} (3.8)$$

to correct the velocity field by

$$u = u^* + u'$$  \hspace{1cm} (3.9)$$

where $p'$ and $u'$ are the pressure correction and the velocity correction, respectively. By subtracting Eq. (3.7) from Eq. (3.6), the relation between $p'$ and $u'$ is obtained as

$$a_e u_e' = \sum a_{nb} u_{nb}' + (p_P' - p_E') A_e$$  \hspace{1cm} (3.10)$$

The exact solution of $p'$ would include pressure corrections and velocity corrections at neighboring points of $u_{nb}$, which would in turn include their neighbors, and so on. The resulting pressure-correction equation will be very complicated, and economically unfeasible to solve. The remarkably successful SIMPLE algorithm of Patankar and Spalding (1972a), which stands for Semi-Implicit Method for Pressure-Linked Equations, provided a simple solution procedure by neglecting the underlined term in Eq. (3.10). However, this results in larger $p'$ values and under-relaxation of both the momentum and pressure correction equations are recommended by Patankar to avoid divergence of the iterative cycle. Such remedy will undoubtedly cause slow convergence rate of the solution.

Van Doormaal and Raithby (1984) modified the SIMPLE method to eliminate the need for under-relaxation of pressure, hence improving the convergence rate. They argued that if $p$ is changed by $p'$, then the velocity at the cell face $e$ changes accordingly by $u'_e$, while the neighboring points change appropriately by $u'_{nb}$. These velocity changes in response to the change of $p'$ will be of the same magnitude. Therefore, the removal of the term $\sum a_{nb} u_{nb}'$ in Eq. (3.10) and the retention of a term of similar magnitude ($\sum a_{nb} u_{nb}'$ appears implicitly on the left-hand side of the same equation) on the left-hand side is considered inconsistent.
A consistent approach is to subtract the term $\sum a_{nb} u'_e$ from both sides of Eq. (3.10) to give

$$(a_e - \sum a_{nb}) u'_e = \sum a_{nb} (u'_{nb} - u'_e) + (p'_P - p'_E) A_e$$

(3.11)

The underlined term is then neglected in order to obtain a simple expression for $p'$. Van Doormaal and Raithby named this approach SIMPLEC with the $C$ appended to the name SIMPLE as an indication that this is a consistent approximation. Applying Eq. (3.9) and the SIMPLEC approximation, Eq. (3.11) becomes

$$u'_e = u'_e + d_e (p'_P - p'_E)$$

(3.12)

where

$$d_e = \frac{A_e}{a_e - \sum a_{nb}}$$

(3.13)

The continuity equation for a finite control volume centered at $P$ under steady state is expressed as

$$(\rho u A)_e - (\rho u A)_w + (\rho v A)_n - (\rho v A)_s = 0$$

(3.14)

Substituting equations like (3.12) for $u$ and $v$ and rearranging, a discretization equation for $p'$ is obtained as

$$a_P p'_P = \sum a_{nb} p'_{nb} + b$$

(3.15)

where

$$a_{nb} = (\rho A d)_{nb}$$

(3.16)

$$a_P = \sum a_{nb}$$

(3.17)

$$b = (\rho u^* A)_w - (\rho u^* A)_e + (\rho v^* A)_s - (\rho v^* A)_n$$

(3.18)

The term $b$ in the pressure-correction equation is essentially the left-hand side of the discretized continuity in Eq. (3.14) evaluated with the guessed velocities. If $b$ is zero, $u^*$ and $v^*$ will satisfy the continuity equation and no pressure correction is necessary. Hence $b$ is a mass source which the pressure corrections must eliminate.

The SIMPLEC algorithm is executed in the following sequence:

1. Guess the pressure field $p^*$.
2. Solve the momentum equation to obtain $u^*$ and $v^*$.
3. Solve the $p'$ equation (3.15).
4. Correct $p$ by Eq. (3.8).
5. Correct $u$ and $v$ by equations like (3.9).
6. Solve other $\phi$ equations (such as turbulence quantities, energy, etc.) and update properties.
7. Treat the corrected $p$ as the new guessed $p^*$, return to step 2, and repeat the whole sequence until convergence is achieved.
3.4 Implementation of Boundary Conditions

3.4.1 Wall Boundary

The boundary conditions are implemented into the computations by modification of the flux transport terms at the cell faces neighboring the boundaries. The following discussion concerns boundary conditions at a western wall and analogous discussion and expressions can be applied to other wall boundaries.

For cells adjacent to the west wall boundary as shown in Fig. 2, no convective flux passes through the west face of the cell which is coincident with the solid wall. A convenient way to treat this condition is to set the term for the total \( \phi \) flux through this cell boundary in Eq. (3.2) to zero. The diffusive flux term \( \Gamma_{\phi} \frac{\partial \phi}{\partial x} \) is transferred to the right-hand side of Eq. (3.2) as a fictitious source term and it is inserted into the finite volume equation via modifications to the linearized source coefficients. For cells adjacent to the western wall boundary, the normal \( P - W \) link is disconnected by setting

\[
a_w = 0
\]  
(3.19)

and the correct expression is inserted into \( S_U \) and \( S_P \). The most common boundary conditions encountered are:

**Boundary Value Specified**

If the value at the boundary \( \phi_B \) is specified, the diffusive flux term is expressed as

\[
- \Gamma_{\phi w} A_w \frac{\phi_P - \phi_B}{\delta x} = -\Gamma_{\phi w} \frac{A_w}{\delta x} \phi_P + \Gamma_{\phi w} \frac{A_w}{\delta x} \phi_B
\]  
(3.20)

Hence the proper linearized terms are

\[
S_U = a'_w \phi_B
\]  
(3.21)

\[
S_P = -a'_w
\]  
(3.22)

where

\[
a'_w = \Gamma_{\phi w} \frac{A_w}{\delta x}
\]  
(3.23)

**Near Wall Value Specified**

If the interior point \( P \) near the western wall boundary is specified to be fixed at a value of \( \phi_F \), then this is easily done by noting that Eq. (3.2) can be reduced to

\[
S_P \phi_P + S_U = 0
\]  
(3.24)

Thus, \( \phi_P \) can be fixed at the value of \( \phi_F \) by defining

\[
S_P = -10^{30}
\]  
(3.25)
\[ Su = \phi_F \cdot 10^{30} \]  

(3.26)

Even though the linkages between \( P \) and the neighboring points \( N \), \( S \), and \( E \) are not broken explicitly, these source terms dominate in the equation for \( \phi_P \) with solution of \( \phi_P = \phi_F \).

**Diffusion Flux Specified**

If the diffusion flux term is specified at the western wall boundary as \( D \), then it is easily seen from Eq. (3.2) that

\[ Sp = 0 \]  

(3.27)

\[ Su = -\frac{D}{V} \]  

(3.28)

If the diffusive flux is also zero, i.e. zero normal gradient, then the setting of \( a_w = 0 \) in Eq. (3.19) is sufficient.

### 3.4.2 Inlet and Outlet Boundaries

All inlet boundary conditions are specified at grid nodes just outside the computational domain or at the computational boundary. For example, consider a top or north wall inlet as shown in Fig. 3(a). The inlet value of the longitudinal velocity is specified at location \( u \), the inlet values of all scalar variables are specified at location \( c \); the inlet value of the transverse velocity is given at location \( v \). Both \( u \) and \( c \) are just outside the computational boundary whereas \( v \) is right at the boundary. Since there are convective and diffusive transports across the inlet, for example cells 1 to 3 in Fig. 3(a), no special treatment is required for cells near the inlet planes.

However, at side-wall or tangential inlets, care has to be taken in the evaluation of the flux contribution from the boundary. Consider the tangential inlet on a west-side wall near the top wall boundary shown in Fig. 3(b). Cell 1 below the tangential inlet should see the solid wall boundary at its north cell face. Therefore the appropriate wall treatment as described in the section above is applied. But cells 2 to 4, which are immediately downstream of the tangential inlet, should have interactions with the scalar and velocity inlet values specified at the \( u \), \( v \) and \( c \) grid nodes. Due to such complication at all the tangential inlets, an algorithm is used to test whether each boundary is a solid wall boundary, top or bottom wall inlet, or tangential wall inlet. An alternative would be to use finer grids near the tangential inlets, but at the expense of higher computational cost and larger storage space.

In the outlet plane the mass flow rate is corrected at the end of each iteration by the overall mass balance. The difference between the total inlet mass and the mass summed over the entire computational domain is distributed evenly to the normal velocity, which is approximated by the upstream value, to give the required mass flow rate and satisfy the law of conservation of mass. No specification of outlet profiles for all other variables is required due to the upwind differencing where the outlet values are not used in the calculations. However, their values at the outlet plane can be approximated by the variable at the nearest upwind nodes.
3.5 Solution Procedure

The set of finite-difference equations and boundary conditions form a system of strongly-coupled simultaneous algebraic equations which are in effect nonlinear since the source terms are functions of many other variables and the momentum equations are strongly linked with the pressure. Beginning with the initial guess of the field, for all variables, the method of solution solves the strongly coupled sets of equations through the inner and outer cycle of iterations.

The inner cycle of iterations solve the set of simultaneous equations for each variable. The technique used here is the well known tri-diagonal matrix algorithm (TDMA). The equations for the grid nodes on each grid line are assembled into a tri-diagonal matrix whereby the values on the neighboring lines are assumed temporarily known by using the most recently calculated values. The tri-diagonal matrix is then solved implicitly using the Thomas algorithm, which is a particular form of the Gaussian elimination method. In the same manner, this procedure is repeated sequentially from left to right along each grid line of the computational domain. This iteration procedure can also be applied to sweep across the domain in an alternating direction fashion. Because values of other variables used in the set of simultaneous equations being solved are only tentative quantities, a few sweeps of the inner iteration is sufficient and full convergence is not necessary.

In the outer cycle of iteration, the set of governing equations are solved sequentially thus modifying each variable in turn while keeping the other variables unchanged. In each outer iteration sequence, the solution procedures follow the steps below:

1. Initialize or guess the fields of all variables both scalar quantities and velocities.
2. Solve the momentum equations to obtain $u^*$ and $v^*$.
3. Solve the pressure correction equation to obtain $p'$.
4. Correct the pressure $p$ and the velocity components by the appropriate equations such as (3.8) and (3.9).
5. Solve the $k$ equation.
6. Solve the $\varepsilon$ equation.
7. Solve the $H$ equation.
8. Solve the $f$ equation.
9. Solve the $m_{fu}$ equation.
10. Calculate $T$, $\rho$, $\mu$ and $\mu_t$.
11. Check for convergence. If not converged, treat the new values of all variables as improved guesses and return to step 2; repeat the process until convergence.
When solving the governing equation for each variable, several sweep may be necessary but the update of the coefficients is not required. In the case of an isothermal flow, steps 8 and 9 will be skipped. If the flow is incompressible, then steps 7 to 9 will be skipped and the evaluations of \( T, \rho \) and \( \mu \) are also not necessary.

### 3.6 Under-Relaxation

For nonlinear problems, under-relaxation is usually used to avoid large changes in the computed variables which may result in divergence in the iterative solution process. At each point, a weighted average of the newly and previously calculated values is determined. The newly calculated value of \( \phi \), at the \( n+1 \) iteration, represented by \( \tilde{\phi}_{n+1} \) is obtained from the finite volume equation (3.4)

\[
ap_p \tilde{\phi}_{p+1} = \sum a_{nb} \phi_{nb}^n + S_U
\]

The under-relaxed value of \( \phi_p \) is given by

\[
\phi_{p+1}^{n+1} = \alpha \tilde{\phi}_{p+1}^n + (1 - \alpha) \phi_p^n
\]

where \( \alpha \) is the under-relaxation factor which is required to be positive but less than unity. For convenience and saving storage space, Eqs. (3.29) and (3.30) are combined to give the direct form

\[
\frac{a_p}{\alpha} \phi_{p+1}^n = \sum a_{nb} \phi_{nb}^n + S_U + (1 - \alpha) \frac{a_p}{\alpha} \phi_p^n
\]

so that the under-relaxed value of \( \phi \) is calculated directly. This is the equation which is actually solved in the computer program. Since the last term in Eq. (3.31) has a positive coefficient, it is cast into the positive part of the linearized source term.

In addition to the dependent variables, other quantities such as density and effective viscosity can be under-relaxed with advantage. For example, the density of a reactive flow can be under-relaxed by

\[
\rho = \alpha \rho_{new} + (1 - \alpha) \rho_{old}
\]

The under-relaxation factors for various quantities are summarized in Table 4. For isothermal flows, no under-relaxation of \( \rho \) is necessary. Since the initial field values for all \( \phi \)'s are usually guessed and far from the correct values, large under-relaxation factors can be used for fast convergence. However, small under-relaxation factors have to be used near convergence to minimize oscillation in the solution. In general, \( \alpha \) for \( u \) and \( v \) are smaller and the rate of convergence is more sensitive to them than to \( \alpha \) of other variables.

### 3.7 Convergence

At the completion of each outer cycle, convergence of the solution is assessed via the residual source criterion. From the discretized Eq. (3.4), the local residual source of the \( \phi \)-equation is defined as

\[
R_{\phi} = \sum a_{nb} \phi_{nb} + S_U - a_p \phi_P
\]
Hence, if the discretized equation is satisfied, $R_\phi$ becomes zero. The sum of the absolute values of all $R$'s over the computational domain is normalized by an appropriate reference value for each $\phi$, and the obtained values are subjected to the test of convergence. The reference value $R_{\phi}^{ref}$ is typically the flux of $\phi$ applied at the boundary such as the total inlet mass flow and momentum.

The solution is assumed to have converged when the normalized value is below a certain fraction $\gamma$ of the reference value

$$\frac{\sum |R_\phi|}{R_{\phi}^{ref}} < \gamma$$

Typically, $\gamma$ is on the order of $10^{-3}$ for flows involving a single species and $10^{-2}$ for flows involving two species. The advantage of this approach is that all the finite volume equations are monitored for proper convergence. With the proper choice of the reference quantities, it can be ascertained that the normalized residuals are all of the same order of magnitude. The criterion that the maximum of all residuals be less than $\gamma$ ensures that all the equations reach convergence.

By monitoring the normalized residual values during the iteration process, the appropriate under-relaxation factor for each dependent variable $\phi$ and other quantities can be determined. If the residuals oscillate or they do not decrease in magnitude, either the under-relaxation factors are not properly chosen or there are some errors in programming which is commonly due to improper specification of boundary conditions. Another possible occurrence is that only some of the residuals converge. This arises as a consequence of a spurious source being introduced to the variable in question, which is also usually due to the improper treatment of the boundary conditions. In other cases, divergence may be due to too few sweeps of the inner cycle of iteration for the quantities in question. The remedy in such situation lies in increasing the number of inner cycle sweep and decreasing the under-relaxation factor for the appropriate quantities.

Low convergence rates can also occur even if no programming error is involved and the proper under-relaxation factors are used. Typically, this occurs if the computational domain is large and the flowfield has multiple inlet streams so that a number of recirculation zones exist. Due to the complexity of the flowfield, an increased number of grid nodes is required to resolve properly all the features in the flow. For such highly elliptic flowfields, low convergence rates can result with the current solution procedure. In some cases, the presumed steady-state solution does not exist. The flow feature may then change from iteration to iteration and the convergence rate may become unacceptably low.

### 3.8 Stability Via False Source

Gosman and Ideriah (1976) have introduced a stabilizing artifice for steady flow problems by the incorporation of a false source term into the discretized equation. If the mass flows do not satisfy the continuity equation, then there may be a situation where all mass flows leave the control volume. Under this circumstance, the finite volume equation becomes singular, for
$a_p$ is given by $\sum a_{nb}$. The remedy is to add a false source $S_f$ to the finite volume equation through the linearized source treatment. The false source is defined as

$$S_f = m_{net} \left( \phi_P^{old} - \phi_P \right)$$

(3.35)

where

$$m_{net} = |m_n - m_s + m_e - m_w|$$

(3.36)

and the mass flows are given by

$$m_{n,s} = (\rho v A)_{n,s}$$
$$m_{e,w} = (\rho u A)_{e,w}$$

(3.37)

The final form of the finite volume equation solved is actually expressed as

$$(a_p + m_{net}) \phi_P = \sum a_{nb} \phi_{nb} + S_U + m_{net} \phi_P^{old}$$

(3.38)

The addition of the false source stabilizes the iterative solution procedure but it has no effect on the final solution. As the solution converges, $\phi_P^{old}$ approaches $\phi_P$ and $S_f$ approaches zero.

### 3.9 Accuracy

The accuracy of a differencing scheme can be judged from the order of the terms retained in the Taylor series expansion. There exists a dilemma that accuracy is associated with higher order scheme whereas numerical stability is achieved with a lower order scheme. The use of upwind differencing eliminates oscillations but introduces false diffusion terms to the difference equations. However, the use of central-differencing significantly reduces this false diffusion but often produces oscillations in the solution that is unrealistic. The hybrid scheme, although yielding physically realistic solutions in all circumstances, introduces excessive numerical (false) diffusion for many two-dimensional flows and all three-dimensional flows because limited grids are employed due to large computer storage requirement and long computing time.

Accuracy of the solution is generally also a function of the number of grid lines used. The approximations of the algebraic expressions used to represent the partial differential equations becomes asymptotically accurate as the grid spacing is reduced. Ideally, for each flow problem, a grid-independent solution can be sought. In practice, this approach to an accurate solution is limited severely by computer storage, time and cost. Another factor affecting the accuracy is the distribution of the grid nodes within the computational domain. The grid should be arranged so that the nodes are concentrated in regions where gradients are high, and reduced in regions where gradients are low. Also, nodal distribution is important whenever streamline curvature exists because of numerical error arising from false diffusion. An approximate expression for the false diffusion coefficient for a two-dimensional situation has been given by de Vahl Davis and Mallinson (1972). The amount of false diffusion can be reduced by employing finer grids or smaller cell Peclet number and, whenever possible, by orienting the grid so that the streamlines are parallel to the mesh. Schemes that give less false diffusion take into account the multi-dimensional nature of the flow by involving more neighbors in the
discretization equation. Several schemes have been reported and tested by Syed, Chiapetta and Gosman (1985) as a proposal to replace the hybrid scheme in the TEACH codes, however, they are significantly more complicated.

A drawback of the present numerical technique based on TEACH-type computer codes is that the irregularly shaped physical geometry has to be discretized to fit the coordinate system using stair steps. The use of stair steps has several consequences. Regardless of physical modeling and numerical accuracy, calculations of wall heat transfer rates can never be accurate since surface areas are not correct. In order to represent the geometry adequately, more computer storage is required. Mesh refinement and alignment with streamline curvature to reduce numerical diffusion is not possible. The calculated flowfield may have been distorted by the incorrect representation of the geometry.

As discussed in section 3.8, for some problems under investigation, the convergence rate is very low. In such cases, considerations of cost and economy may result in termination of the solution before convergence criterion is reached. An arbitrary limit is then set on the accuracy of the solution by considerations of computing cost.

Boundary conditions and locations of the boundaries both affect the accuracy of the solution field. Improper specifications and treatments of conditions at boundaries may result in divergence, instability or incorrect converged solution.

Finally, the adequacy of the turbulence model used affects the accuracy and stability of the predicted solution. Evaluation of various turbulence models as applied to combustor flows has been reported by many authors such as Srinivasan et al. (1983), Nallasamy (1985), and Nikjooy and So (1987).

3.10 Successive Runs

Since combustor flow is far more complicated than most basic flows, it is not recommended that the inlet velocity of each louvre or jet inlet be inserted at its full strength in starting the computation because the solution is most likely diverging even with very small relaxation factors. To obtain convergence, the computation should be started with small inlet velocities. Other boundary values can be prescribed at their maximum values from the very beginning. As the solution converges to a certain limit, the inlet velocities can be further increased. This gradual increase of inlet velocity procedure is very efficient in obtaining convergence.

A piece of long straight pipe is added to the end of the combustor geometry to ensure that no recirculation exists at the exit plane of the computational domain. To minimize false diffusion and to be able to simulate the dimensions of the louvres and jet inlets as accurately as possible, fine equally-spaced grids are used in the combustor flowfield, and unevenly-spaced grids are used in the exhaust piping. The number of grids employed in the code is 220 x 53. The code was run on a Gould 32/9705 computer system and on the CRAY X-MP/22 at the University of Toronto. It took 1.73 and 2.58 seconds per iteration on the CRAY for the calculations of cold flow with single component and five components of chemical species,
respectively. The residual terms are less than 1\% at convergence.
Chapter 4

Experimental Set-up

4.1 Combustor Rig

The experimental data presented in this investigation was obtained from the sector-combustor rig shown in Fig. 4. The test section contained a straight 90° sector of a compact Pratt & Whitney Canada toroidal-vortex reverse-flow annular experimental combustor PW209T. The sector combustor was designed so that the width of the top liner would be larger than the bottom liner so that the radial variation in flow area was accounted for. The cross-section of the sector combustor is shown in Fig. 5. The locations of louvres, jet inlets, front dome and exit plane are indicated in the schematic diagram. There are a total of three different sets of jets and nine sets of wall louvres. One set of wall louvres is the thumb-nail louvre which directs the air flow in the tangential direction. Two inclined side windows mounted flush to the combustor allowed optical access to the primary and main part of the secondary zone for laser Doppler velocimetry (LDV) and flow visualization. In the cold flow without-fuel-injection experiments, the ignitor and fuel injector ports were sealed. Figure 6 shows the air supply circuitry. Pressure regulators and a rotameter were used to control the amount of air entering the sector rig. Downstream of the main air rotameter, the compressed air entered a 7.37 cm diameter pipe, which led into a 3.18 cm by 23.81 cm rectangular area change inlet duct and the particle seeding section before entering into the inlet port of the combustor rig. A secondary circuit formed by diverting part of the main air flow was used to seed the flow with particles for LDV measurements. At the exhaust end of the rig, the reverse-flow exhaust ducting was removed and a rear window was installed allowing full end-view of the sector-combustor. The exhaust plenum was ducted vertically to the outside ambient. The entire rig was mounted on a computer-controlled three axis motion table which could be moved 0.4283 m, 0.3026 m, and 0.3073 m in the x, y, and z direction, respectively. The travel resolution of each axis was 3.175 μm in the x and y directions and 0.635 μm along the z direction. To check the position of the control volume, dial gauges were mounted on all three orthogonal axes of the table with spatial resolutions of 25.4 μm in the x and z directions, and 10 μm along the y axis.

To simulate certain operating conditions of the combustor, the mass flow through the sector may be scaled according to the parameter $mT^{1/2}/AP$. In both the cold and hot flow
experiments, a constant 204 mm (8 in.) of water static pressure drop across the combustor was maintained. The air inlet temperature was 23°C. A water manometer and the main flow rotameter were used to monitor the flow conditions. If the flow through the main rotameters dropped by 3% the experiment was stopped and the combustor was cleaned. The drop in the air flow rate was caused by the build-up of seeding particles immediately downstream of the louvres and the clogging of the tiny louvre inlet holes. The combustor was cleaned typically after every two and a half hours of operation. For runs with fuel injection, a modified Parker-Hannifin pure air-blast fuel nozzle was used at $0^\circ$ orientation. The fuel nozzle has three annuli where swirling air was ejected from the outer annulus and co-swirling fuel was injected from the inner annulus. A second fuel line delivered fuel to the center core where it is premixed with air and then injected into the combustor through the center core. An ignitor was axially mounted on axis with the fuel injector. Commercial grade methane gas of 96% purity was used instead of liquid fuel in order to avoid fouling up of the windows. The fuel flow rate was set under the condition that the flame did not scavenge the side windows. The fuel-air ratio employed for all fuel-injection runs was 0.004 or an equivalent fuel-air ratio of 0.005 with kerosene.

The ignition characteristics were obtained by varying the air and fuel flow rates. For ignition, the fuel-air mixture was chosen to be twice as rich as in normal operating conditions. The fuel flow was fixed throughout ignition and running conditions. Only the air flow rate was reduced by 50% for ignition and then gradually increased to 100% once the flame was established. The ignition was very repeatable and met the 10 second time-to-light (TTL) ignition criterion.

### 4.2 Laser Doppler Velocimetry and Data Acquisition System

The measurement of turbulent recirculating flows has been difficult to perform with the conventional measuring techniques, such as pitot tubes and hot-wire anemometry, due to their inability to distinguish a change in flow direction and to adequately register flow fluctuations in excess of 30% of the local mean velocity. They also share the disadvantage of disturbing the flow. With the advent of laser Doppler velocimetry, the measurement of complex turbulent isothermal and combusting flowfields are made easier. Some of the advantages of LDV are: it is a linear, non-intrusive method with good spatial resolution, it has the capability of determining the instantaneous flow direction and the simultaneous measurement of velocity components and the corresponding turbulence intensities. Some of the disadvantages or difficulties of LDV are discussed later.

#### 4.2.1 Principles of LDV

The laser Doppler velocimetry technique uses the Doppler frequency shift effect of the light scattered by moving particles to determine the particle velocity. If the particle size is properly
chosen, the particle will follow the flow trajectory and hence the flowfield velocity is known.

Consider the differential Doppler or dual-beam mode used in the present setup where two light beams of frequency \( f \) are directed onto a small particle which is moving with velocity \( V \), as shown in Fig. 7(a). Assume that these two beams are of equal intensity and coherent. The unit vectors of the two incident laser beams, \( \hat{e}_1 \) and \( \hat{e}_2 \), depend on the geometry of the system. The general equation expressing the Doppler frequency shift \( f_d \) of the scattered light as a function of particle velocity \( V_p \) is

\[
f_d = |f_{s1} - f_{s2}| = \left| \frac{V_p}{\lambda} \cdot (\hat{e}_1 - \hat{e}_2) \right|
\]

where \( f_{s1} \) and \( f_{s2} \) are the frequencies of the scattered light and \( \lambda \) is the wavelength of the laser beam. The velocity component \( u \) of the particle velocity \( V_p \) measured is in the plane formed by \( \hat{e}_1 \) and \( \hat{e}_2 \) and normal to the bisector line of the angle \( \theta \) formed by the two unit vectors.

### 4.2.2 Fringe model

The basis of the differential Doppler mode in LDV is also commonly explained by the fringe model. The two laser beams of monochromatic coherent light with plane and parallel wavefronts form a system of parallel alternately dark and bright fringe pattern at the beam crossing (control volume) due to the interference of the wavefronts as shown in Fig. 7(b). The distance between the fringes \( \delta_f \) can be calculated from the angle of intersection \( \theta \) and the wavelength of the laser beam \( \lambda \)

\[
\delta_f = \frac{\lambda}{2 \sin (\theta/2)}
\]

The velocity component \( u \) of the particle normal to the fringe planes will scatter light at a frequency

\[
f_d = \frac{|u|}{\delta_f}
\]

or

\[
f_d = \frac{2 |u|}{\lambda} \sin (\theta/2)
\]

The Doppler frequency of the scattered light \( f_d \) given by Eqn. (4.1) or (4.4) implies several significant points of differential Doppler operation mode. The most important implication is that the observed signal frequency is independent of the direction of the receiving system. Hence the receiving optics can be located where the scattered light intensity is at a maximum, which in turn increases the signal to noise ratio. The relationship between the Doppler frequency \( f \) and the scattered light \( f_d \) obtained at the output of the photomultiplier and the velocity component normal to the angle bisector of the laser beams is linear. This is a significant advantage of LDV over the other velocity measuring devices (e.g. hot wire anemometry), which contributes to the high accuracy of the technique. Also, the frequency \( f_d \) is determined by the normal velocity component \( u \) only and is entirely insensitive to other velocity components. The signal frequency \( f_d \) to be processed can be appropriately adjusted by the intersection angle \( \theta \). However, another device is necessary to determine the sign of the velocity component since \( +u \) and \( -u \) will provide the same signal frequency.
4.2.3 Transmitting Optics

The simultaneous measurement of the two mean velocity components and their corresponding turbulence intensities were obtained with a 4-beam, 2-color Argon-ion LDV system operating at 2W continuous power in the dual-beam forward scatter mode. The optical arrangement was built up from standard DISA 55X Modular LDA Optics components as shown in Fig. 8. A Spectra Physics model 171-07 Argon-ion laser provided the coherent light source which has very high frequency stability. The laser was operated in the fundamental transverse electromagnetic mode TEMₐ₀, giving a beam with a Gaussian light intensity profile and is characterized by a beam waist (the narrowest section of the laser beam), in the vicinity of which the electromagnetic wave system is planar. The two focused Gaussian laser beams must intersect at their waists in the control volume in order to obtain interference fringes that are equidistant throughout. Otherwise, a set of diverging fringes will result in the control volume, and Doppler signals of different frequencies will be produced when particles of the same velocity pass through the control volume at various locations; generating, therefore, erroneous data.

A perfect fringe system can be produced only if the polarization characteristics of the two intersecting laser beams are identical. To enable the polarization of the beams to follow the direction of rotation and maintain the same quality of light intensity when the Modular LDA Optics is rotated, the 55X20 retarder and cover were used. The laser beam first passed through a λ/4 retarder plate mounted on the laser head to convert the linearly polarized laser beam into a circularly polarized beam. The beam then passed a second λ/4 retarder plate mounted on the Modular LDA Optics to convert the circularly polarized beam back into a linearly polarized laser beam with correct polarization direction relative to the optical axis of the Modular LDA Optics.

To adjust the beam waists of the two laser beams at their intersection at the control volume when a front lens with long focal length is used, the 55X22 beam waist adjuster is positioned in the laser beam between the second retarder and the cover. The Gaussian laser beam is transformed into a smaller beam waist which could be positioned at an arbitrary location as desired. Proper positioning of the beam not only eliminates the undesirable variation in fringe spacing, but it also increases signal to noise ratio significantly because the minimum beam diameter and maximum light intensity are centered in the control volume.

The laser beam then enters the 55X25 beam splitter where the incoming beam is split into two parallel beams with equal light intensity (50:50 ratio). Frequency shifting of the laser beam is essential in measuring flow directions and eliminating fringe bias. Without frequency shifting, the fringes in the control volume are stationary and the system is unable to detect which direction the particle being measured is moving. The frequency of the central beam light wave is optically upshifted by 40 MHz relative to the other beam with the 55X29 Bragg cell. The upshifted beam is passed through a glass rod to ensure that it has identical optical path length as the shifted beam to maintain coherency. After the Bragg cell, the 55X27 modified color beam splitters separate both the shifted and unshifted incoming two color beams into a green and a blue beam each, which are all parallel to the Modular LDA Optics axis. The wavelength of the blue and green beams are 488 nm and 514.5 nm. The four parallel beams pass through the 55X30 backscatter section and the 55X31 pinhole sections which are intended
for backscattering operation mode (not used in the present investigation). The four beams then enter the 55X32 modified beam translator where the outgoing beam separation can be varied between 9.2 mm and 27.6 mm. The fringe spacing separation and the fringe number \( N_f \) are in turn modified depending on the beam separation distance.

It is common to reduce the diameters of the beams in the control volume to obtain a significant increase in the light intensity. The 55X12 beam expander with an expansion ratio of 95:50 is used. Since reduction by a factor of two in the focused beam diameter results in an increase of a factor of four in the light intensity, the signal level can be improved significantly.

Last in the transmitting optics is the 55X58 achromatic front lens with a focal length of 600 mm. This front lens causes the four beams to intersect at an angle \( \theta = 6.87^\circ \) and form the control volume. The spacing of the interference fringes in the control volume were 4.074 \( \mu \)m and 4.295 \( \mu \)m for 488 nm and 514.5 nm wavelengths, respectively. The dimensions of the probe volume were 2.15 mm on the major axis and 0.128 mm on the minor axis. Hence there are two distinct control volumes intersecting each other with different waists dimensions. To obtain measurement of the two velocity components at the exact same point in space, these two control volumes have to be properly focused.

### 4.2.4 Receiving Optics

Light is scattered at the two wavelengths (488 nm and 514.5 nm) as a seeding particle passes through the control volume. The scattered light is then collected in the forward on-axis direction and focuses onto the 55X34 photomultiplier optics with a 0.1 mm pinhole diameter and a variable focus from 600 mm to infinity. The focusing of the objective on the control volume can be easily done with the aid of an eyepiece and a crosswire. The pinhole acts as a spatial filter which removes unwanted stray lights scattered from the area near the control volume. The collected light then passes through the 55X35 color separator with a color separating beamsplitter prism which separates light into 488 nm and 514.5 nm wavelengths (blue and green color). In front of each 55X08 photomultiplier section is an interference filter 55X36/55X37 (blue and green) which removes reflections, undesirable non-laser light (such as ambient lights and luminous flames) and cross-talk from the other laser beam. With these narrow bandwidth 10 nm filters, the signal to noise ratio and the channel separation of the blue and green lights are increased. The photomultiplier section then amplifies the signal voltage, which is then sent to the LDV signal processor.

### 4.2.5 Particle Seeding

Most of the measuring problems with LDV stem from the use of seeding particles. For the particles to follow the flow and the velocity fluctuations properly, they must be small (less than 1 \( \mu \)m) in size. In general, the higher the flow velocities and frequencies of the turbulent fluctuations, the smaller the particles must be in size. At a higher velocity, the transit time of the particles through the control volume is small and fewer photons are scattered for the same laser light intensity which results in a reduction of the signal to noise ratio. Also, the
frequencies to be processed are increased and the bandwidth of the counter processor must be enlarged. Since the signal to noise ratio is inversely proportional to the bandwidth, increasing the bandwidth for higher velocities reduces the signal to noise ratio and the accuracy of the result.

During the experiment, the entire flow domain was sparsely seeded with micron and sub-micron size particles so that approximately one particle was present in probe volume at a given instant of time. The particle seeder used in these experiments, as shown in Fig. 9, was made up of an 8 cm inner diameter cylinder with four diametrically-placed air injectors mounted near the base. Above the injectors, a stretch piece cloth separated the fluidized bed from the injector chamber. The fluidized bed consisted of a mixer of sub-micron size MgO and 170-200 μm glass beads. This mixer was used in the cold flow without-fuel-injector experiments. Sub-micron Al₂O₃ particles were used instead of MgO in with-fuel-injection experiments. The glass beads were used to minimize agglomeration. Air from the injectors would fluidize the mixer and the sub-micron size particles would be suspended and collected by an air ejector situated at the top of the cylinder. Large agglomerated particles were electronically removed by the amplitude limiter on each of the TSI counters. The average data rates ranged from counts of 1000/s to 5000/s, depending on the zone measured. The data rate was around 1000/s within the recirculation zone, 3000/s and higher near dilution jets, and around 2000/s in the rest of the flow.

4.2.6 Signal Processing

Flows having a low seeding particle density will typically produce a Doppler signal with short bursts followed by no-signal intervals. The counter processors operate best with only one particle at a time in the control volume, although they will also work for nearly continuous bursts. Some of the advantages of counter systems are their high accuracy, no limitations of the particle rate on the low end, high reliability of the results due to the inclusion of the validation circuits, high frequencies of up to 100 MHz or more can be measured, and the signal to noise ratio of the acceptable signals can be appropriately set with the amplitude limit.

An oscilloscope was used to observe the traces of the Doppler signal under high sweep speed. The amplified signals from the photomultipliers were fed into two TSI Model 1980 counter type signal processors. The incoming signals were first filtered by the high pass and low pass filters. The high pass filter removed the pedestal or the dc component from the signal, while the low pass filter removed the high frequency noise from the signal. These filters were set to encompass the frequency range of the signal to be measured. The dynamic frequency range could be observed from the plot of the velocity probability density function PDF. The gain of the input section was adjusted to obtain proper signal amplitude which was indicated by a green light.

Large seeding particles typically produce a higher amplitude pedestal than small particles. To reduce the chance of accepting large particles that might not follow the flow, the maximum pedestal amplitude that would be accepted was appropriately set by the amplitude limit. The filtered or conditioned frequency burst was then used to gate a high frequency clock over
a fixed number of Doppler periods or cycles of $N = 16$. The data acquisition process began as soon as the Doppler signal exceeded the Schmidt trigger voltage level. The zero crossing detector then produced a rectangular wave for each positive going of the signal. A first counter then gated the time period of a high speed clock of 250 MHz over $N/2$ cycles, and a second counter gated the time period over $N$ cycles. A comparator circuit then compared the first count with $1/2$ of the second count. The signal was then accepted for future processing only if the comparison was less than 1%. The maximum time of the gated signal to be accepted was set to auto ranging mode. The mean error determining the time interval is $\pm 1/2$ clock cycle or 2 ns since the best significant bit is a measure of whether the clock was high or low when it was gated off.

The counters output the frequency of a validated burst in the digital form of 12 bit mantissa, 8 bit cycle number and 4 bit exponent. Hence the Doppler frequency was calculated from the equation

$$f_d = \frac{N \cdot 10^9}{D_m \cdot 2^{n-2}} - 40 \text{ MHz}$$

where $N$ is the number of counted cycles, $D_m$ is the digital mantissa and $n$ is the exponent. The 21 bits of digital information from each counter was fed into a Motorola 32-bit microcomputer if the processed signals fell within the 16 $\mu$s coincidence time window. A coincidence window was required to ensure that the Doppler burst from both counters was due to the same particle. To minimize velocity bias, the microcomputer was programmed to accept data from the counters at equal time intervals.

### 4.2.7 Data Reduction

When both counters released a *data ready* signal, information from the counters was transferred to the computer. The computer in turn sent an inhibit signal to disable the counter until data reduction in the computer was complete. The evaluation of the mean velocity components, the normal stresses and shear stress were obtained using the following formulas:

$$\bar{u} = \frac{1}{N} \sum_{p=1}^{N} u_p$$  \hspace{1cm} (4.6)

$$\bar{v} = \frac{1}{N} \sum_{p=1}^{N} v_p$$  \hspace{1cm} (4.7)

$$\bar{w}^2 = \frac{1}{N} \sum_{p=1}^{N} (u_p - \bar{u})^2$$  \hspace{1cm} (4.8)

$$\bar{v}^2 = \frac{1}{N} \sum_{p=1}^{N} (v_p - \bar{v})^2$$  \hspace{1cm} (4.9)

$$\bar{u}'v' = \frac{1}{N} \sum_{p=1}^{N} (u_p - \bar{u})(v_p - \bar{v})$$  \hspace{1cm} (4.10)
where \( N \) was the number Doppler burst accepted. \( N \) usually was \( \sim 10000 \) for most of the test section. But within the high shear zone, \( \sim 50000 \) burst were measured.

With the validated \( N \) number of realizations, it is customary to exercise control over them in order to be highly confident that the evaluated results are close estimates of their true values. It is a common practice to use \( \pm 3\sigma \) as the upper and lower control limits of the samples, where \( \sigma \) is the standard deviation defined as (e.g. for velocity \( u \))

\[
\sigma = \left[ \frac{1}{N} \sum_{p=1}^{N} (u_p - \bar{u})^2 \right]^{1/2}
\]  

(4.11)

Realizations that fell outside the \( \pm 3\sigma \) limits of the mean value were removed. Then the mean velocities and stresses were recalculated. The frequency range output to the counter processor depended on the mean and standard deviation of the measured velocity component (\( \bar{u} \) and \( \sigma \)), the fringe spacing \( \delta_f \), and the frequency shift \( f_s \). With the assumption that the velocity fluctuation distribution was Gaussian, and applying \( \pm 3\sigma \) upper and lower control limits to the samples, the maximum and minimum frequencies estimated would be

\[
\begin{align*}
f_{\text{max}} &= \frac{\bar{u}}{\delta_f} \left( 1 + \frac{3\sigma}{\bar{u}} \right) + f_s \\
f_{\text{min}} &= \frac{\bar{u}}{\delta_f} \left( 1 - \frac{3\sigma}{\bar{u}} \right) + f_s
\end{align*}
\]  

(4.12) (4.13)

The velocity probability distribution function (PDF) was an option that could be plotted on the computer screen. From the PDF one was able to determine if the filter limits should be adjusted. If the PDF distribution was such that some of the high or low velocity particles were being filtered out, the filter settings would be widened.

### 4.3 Flow Visualization

Visualization of the flow without fuel injection was performed with the Argon-ion laser operations at 8 W, all-lines and a 4.8 mm diameter glass rod mounted behind the rear combustor window. A cross-section of the combustor was illuminated by a sheet of light when the laser beam passed through the glass rod. Glass spheres were injected into the flow at the particle seeding section. A video camera was used to record the Mie scattered light from the glass beads as they passed through the sheet of light. Three size ranges of glass beads were used: 1-37, 37-57 and 57-80 \( \mu \)m. The larger glass beads had the advantage of producing more intense scattered light but had the disadvantage of not being able to follow the flow. Hence, the 1-37 \( \mu \)m beads gave the qualitative picture of the flow.

In the hot flow experiment, the combustion of methane gas gave an easily visible bluish-white flame. The longitudinal and transverse spread of the flame was viewed from the side windows, while the transverse and tangential spread of the flame was viewed from the rear window. A video was made of both the side and rear view of the flame.
4.4 Accuracy of Measurements

Although the use of LDV as a diagnostic tool in the measurement of combustor flow is preferable over other intrusive techniques, there are many factors that influence the accuracy of LDV measurements. Described below are some of the problems that are commonly encountered in LDV measurements.

4.4.1 Control Volume

The laser and the optics were initially aligned without any windows. But when the four beams passed through the side window, the control volume was deflected downwards and the four beams did not focus at a common point. A microscope objective was placed inside the combustor so that the control volume image was displayed on a screen. In this way the beam translator prisms were adjusted so that all four beams intersected at the control volume. The position of the control volume was set by etching a centering mark at the rim of the top liner. Then the combustor was moved until three of the beams appeared to intersect at the etched mark. A second way to ensure the center-plane position of control volume was to traverse the upper dilution jet, and the y position where the velocity vector was maximum was considered to be the center-plane of the combustor. Before the mapping of each plane, the centerline velocities and stresses of that plane were measured. Thus whenever the control volume needed to be realigned due to drift, the control volume was repositioned so that the planer centerline velocities and stresses were within 5% of the original values.

4.4.2 Statistical Bias

Inherent to LDV measurements in turbulent flows are a number of statistical biases which must be minimized to obtain reliable data. Discussed next are the different statistical biases and the methods use to minimize them.

Velocity Bias

Velocity bias is a consequence of the particle arrival rate at the control volume to be statistically dependent on the instantaneous flow velocity. In other words, statistically, more high velocity particles will pass through the control volume than low velocity particles over a given time interval. If the stored data rate is the same as the counter data rate (validation rates), the velocity would be statistically biased high when the arithmetic mean is used.

During the Second International Laser Anemometry Symposium at Miami Beach, a panel of experts met to discuss several processing methods for the elimination of velocity and other statistical biases (Edwards, 1987). One of the methods discussed was the controlled processor approach (Erdmann and Tropea, 1981 and Edwards, 1981) which was used in the present experiments. This approach divided the total measurement time into N (number of samples) equal time intervals. The first validate signal that appeared in each time interval was stored. The sampling rate chosen in these experiments was 1/20 of the lowest data rate of the two
counters. The means of the velocities, and stresses were determined from the arithmetic mean. The saturable detector method (Simpson and Chew, 1979; Roesler, Stevenson and Thompson, 1980; Stevenson, Thompson and Luchik, 1982; Stevenson, Thompson and Gould, 1983; Edward and Jensen, 1983) discussed also in the paper is different from the controlled processor approach in that the interval time is stopped until the first valid signal has been accepted and then the interval timer is restarted. As with the controlled processor method, by decreasing the sampling rate the saturable detector method approaches constant time sampling which produces unbiased data.

Craig, Nejad, Hahn and Schwartzkopf (1986) investigated the highly turbulent flowfield of a ramjet dump model combustor and established a criterion for the ratio of seeding to sampling rate to yield unbiased data. Using the saturable detector method, a maximum collection time error of 1%, they indicated that a seeding to sampling ratio of 10:1 is necessary to unbias the data in a flow region where the local turbulence intensity is 50%. As previously mentioned, the present experiments used a sampling ratio of 20:1.

Other Biases

The two primary reasons for filter bias are incorrect counter filter settings and the photodetector efficiency being frequency dependent. If the filter limits were set too narrow, a scattering particle might generate a signal frequency which was outside the filter bandwidth and thus the counters might not process the signal. Observation of the ±3σ upper and lower control limits on the PDF plot would indicate if the filter settings should be adjusted. To minimize the effect of decreased photodetector efficiency at higher frequencies, the laser was run at 2 W and the photodetector and counter gains were increased to minimize the data rate while maintaining good quality Doppler burst, as observed on an oscilloscope.

When a particle passes through the control volume parallel to or at a small angle with the fringes, it may not cross enough fringes to be processed by the counter. This is known as angle or fringe bias. The popular method to eliminate this bias is to frequency shift one or both of the beams. The effective fringe velocity should be at least twice the maximum Doppler frequency to give a uniform angular response. A Bragg cell operating at 40 MHz was used to minimize fringe bias in the flows presented in the present work.

Gradient bias occurs when there exists a velocity gradient across the finite size control volume. This bias may become important when measurements are taken across thin shear layers where different velocities can be present in the finite control volume at the same time. Note that gradient bias is not an error caused by velocity fluctuation. Gradient bias may be minimized by increasing the intersection angle so as to decrease the control volume size, collecting the scattered light off-axis to avoid stray-light from entering the receiving optics and hence increase the signal to noise ratio, and by arranging the control volume minor axis along the direction of the velocity gradient to obtain similar effect as to reduce the size of the control volume.

Combusting flows may produce non-uniform seeding due to fluctuations in fluid density, which results in a biasing known as density bias. The inherent inhomogeneous seeding due to thermal expansion of combustion products is another problem typical in combusting flows.
This thermal expansion has the tendency to decrease the volumetric particle concentration because a volume of cold gas containing a number of particles expands to a larger volume of hot gas with the same number of particles. The use of constant time-interval sampling, as suggested by Simpson and Chew (1979), will not only minimize velocity biasing, it will also minimize density biasing.
Chapter 5

Combustor Flow Without Fuel Injection

Since the combustor inside the rig is only a sector of a full annular combustor, the terminology referring to the liners will be different than the terms normally used for an annular combustor. The outer liner will be called the top liner while the inner liner is called the bottom liner here.

5.1 Flow Visualization

The flowfield of a cross-section of the investigated sector combustor was made visible by densely seeding the flow with submicron glass beads and illuminating them with a sheet of laser light in the green spectrum. The video-taped pictures showed clearly the characteristics of the complex flowfield inside the complex combustor geometry. Due to the contraction of the combustor near the exit, the sheet of laser light was unable to illuminate a small portion of the flowfield in the primary zone adjacent to the top liner.

In the primary zone, an oval-shaped recirculation zone was found rotating in a counter-clockwise direction, with its center skewed upward and downstream of the zone. This primary recirculating flow is also commonly known as the primary vortex. In a full annular reverse-flow combustor, this primary vortex extends all around circumferentially, and thus is called the toroidal vortex which has the shape of a donut. This recirculating flow occupied a large region of the entire flowfield (about 50% of the test section). In the case of combustion, it is this recirculation zone that provides the heat to sustain combustion by means of upstream convection, and the turbulent mixing of the unburnt fuel and air stream with the hot combustion products. Also in this zone, the flow velocity was lower near the center of rotation than away from it. It is this velocity reduction that provided the necessary shelter for flame stabilization. This recirculation zone was formed when an adverse pressure gradient existed in the flow which surmounted the momentum of the normal flow. This was brought about by the rotational flow motion setup by the side wall louvres and further enhanced by the deflection jet.
The primary jet, which was located at the bottom liner in the primary recirculation zone, could be clearly seen. The vertical jet was bent downstream by the strong crossflow with the primary recirculation. The purpose of the primary jet is to increase the intensity of and confine the primary recirculation to the primary zone. A smaller, but more intense recirculation zone flowing in a clockwise direction was also observed. This recirculation zone was the result of the low pressure region which was created by the high velocity primary jet, and it is also commonly known as the secondary recirculation zone.

Further downstream of the secondary recirculation zone were the top and bottom dilution jets. The top dilution jet is on the same plane as the primary jet. Hence when visualizing the flow of the primary jet, the top dilution jet was also seen. Its velocity and momentum were so large that it was seen to be penetrating right across the flowfield, as it was expanding outward, and impinging on the bottom liner. The bottom dilution jet could be visualized by moving the three-axis traversing table in the $y$ direction to a different plane. Alike the top dilution jet, the bottom dilution jet was also found to be of high velocity and momentum. It expanded as it penetrated across the flowfield and impinged against the top liner. Downstream of the dilution jets, the flow was seen to be exiting almost horizontally.

### 5.2 Experimental Results

At the beginning of the mapping of the flow properties in a plane, the centerline properties in that plane were first measured and used as reference data. During the experiment, the data at the centerline would be checked with the reference values. If a discrepancy of more than ±1 m/s was found in either the $\bar{u}$ or $\bar{v}$ velocity, the measured data of the entire grid line would be discarded. Since the experimental setup was for 2-D measurements, the turbulence kinetic energy $k$ was taken as $(\bar{u}^2 + \bar{v}^2)/2$ in the analysis below. Measurements of the mean velocity components, and mean turbulence intensities provide quantitative information on the complex flow structure inside a real combustor geometry. Experimental results are presented in vector plots, 2-D contour plots and 3-D surface plots. Because of high gradients that exist near the wall for most measured quantities and the surface plot software attempts to extrapolate data outside the wall boundaries, the gradients near the wall may be attenuated and in some cases slight oscillations may be present. In the surface plot, the horizontal plane drawn outside the flow domain is the zero reference plane.

Three sections of the combustor were investigated under cold flow without fuel injection. The $\bar{u}$ and $\bar{v}$ components of the mean flow velocity, the normal stresses $\bar{u}^2$ and $\bar{v}^2$, and the shear stress $\bar{u}'\bar{v}'$ were mapped with equal spacing of 2.54 mm (0.1 inch.) in both directions within a section. Experiments were done on three separate planes on the $y$ axis as shown in Fig. 10: a) Top-Jet Plane, b) Inter-Jet Plane, and c) Bottom-Jet Plane. In the sketch, not all the jet inlet holes are shown. The primary and bottom jet holes, which are located in the bottom liner, are drawn in lighter lines. At the center plane of the sector-combustor, the top dilution hole was in alignment with a deflection hole. This was used as the reference plane and was referred to as Section 1 (Top-Jet Plane). The second plane chosen was located 3.05 mm away from Section 1. Since this plane was situated between the top and bottom
dilution holes, it was referred to as Section 2 (Inter-jets Plane). The third plane studied was 5.89 mm away from Section 1 where the nearest bottom dilution hole was located, and this was referred to as Section 3 (Bottom-Jet Plane). Near the boundary of the liner walls, only the control volume of one component of the LDV system could penetrate into the flowfield. Therefore near the boundaries, the laser optics were rotated to obtain data at two different angular positions (sensitivity vectors). The $\bar{u}$ and $\bar{v}$ components were then obtained from the evaluation equation for a one-dimensional optical unit. For the no fuel-injection runs, only the velocities and one of the normal stresses were obtained near the liner walls. A reference point was chosen on the combustor for the description of measurement point and the location of the control volume inside the test section. This reference point is chosen to be the rim of the top liner in the front dome as shown in the contour plots below with an asterisk.

5.3 Flow Velocities

5.3.1 Velocity Vectors

Section 1 (Top-Jet Plane)

Figure 11 depicts the measured velocity vectors in Section 1 (Top Jet Plane). The length of a vector represents the magnitude of the velocity and the arrow head shows the corresponding flow direction. The measured velocity field in the primary zone indicates that the primary zone recirculation rotates in a counterclockwise direction. The center of rotation is located at $(3.683 \text{ cm}, 1.143 \text{ cm})$ from the reference point with velocity near zero. The magnitude of the flow velocities decreases from the liners toward the center of the recirculation zone. Because of such velocity profile, the primary vortex has the characteristic of a solid body rotation. The velocity at the wall louvres was extrapolated from the nearest measured values to be about 15 m/s. The primary jet has a measured maximum velocity magnitude of 25 m/s with a trajectory angle of 60° near the jet exit. The secondary recirculation zone immediately downstream of the primary jet rotates in a clockwise direction in an elliptical shape with the major axis tilting at an angle of about 55° from the horizontal plane. The top dilution jet has a maximum measured velocity of 47.4 m/s at an angle of $-114°$ below the horizontal plane. This top dilution jet has such a high momentum that it penetrates across the flowfield and strikes the bottom liner with a velocity of about 13.5 m/s. The flow near the top liner downstream of the top dilution jet exits at quite a low velocity of about 7 m/s horizontally. The three-dimensional nature of the crossflow is seen on the downstream side of the bottom dilution jet (not located in this plane) near the bottom liner. The low velocity region below the centerline is part of the wake flow from the bottom dilution jet. Some error is found in the measurement of a few lines just upstream of the primary jet (traverse lines 12, 14 and 15), likely due to drifting of the control volume away from the measuring plane. The maximum deviation of $\bar{u}$ and $\bar{v}$ from the original centerline sweep is $\pm 1$ and $\pm 2$ m/s, respectively for these three lines. These errors are not very obvious in the velocity plots but are more noticeable in the plots of the Reynolds stresses. Since the velocity is generally small in the primary zone which is in big contrast with the dilution zone, the error is not visible in the surface plots.
Because these errors were observed after the combustor rig had been returned to PWC, no remeasurements were performed.

Section 2 (Inter-Jets Plane)

Figure 12 depicts the measured velocity vectors in Section 2 (Inter-Jets Plane). The recirculation in the primary zone becomes more of a circular than an oval shape as found in the Top-Jet Plane. This is the result of the absence of the primary jet hole in this plane. However, the spread of the deflection jet still influences the flow pattern, but to a lesser extent. The vortex core is very much elongated and a likely center is located at (2.54 cm, 1.52 cm) from the reference point. The influence of the primary jet can be seen from the sudden increase in velocity and change of flow direction in the primary jet location, although the maximum measured velocity here is only 15.6 m/s. The trajectory angle of the primary jet has decreased to 40° above the horizontal plane (compare with 60° in the Top-Jet Plane). The secondary recirculation zone immediately downstream of the primary jet in this plane also has its major axis of rotation reduced to about 40° above the horizontal plane. The center of this secondary recirculation zone is not as clearly defined as in the Top-Jet plane. Further downstream of the secondary recirculation zone near the top liner of the dilution zone, the main flow from the primary zone is slightly disturbed by the top dilution jets. The spread and penetration of the top dilution jet is shown clearly immediately behind the secondary recirculation zone with still fairly strong negative velocity components in the transverse direction. The spread of the bottom dilution jet is also found in the dilution zone interacting with the top dilution jet before turning towards the exit plane. The impression of the two sets of opposing dilution jets on the flowfield is not expected to be equivalent. Three flow features are responsible for the offset of this symmetry. First is the primary jet and the secondary recirculation that it creates. The primary jet introduces high mass flow velocity towards the top liner, which subsequently flows along the upper half of the dilution zone towards the exit plane and reduces the penetration of the bottom dilution jet. Second, the secondary recirculation is responsible for the splitting up of the primary flow and drawing a part of the primary flow towards the bottom liner. This also contributes to the suppression of the bottom jet's strength in the flowfield. Third, the opposing dilution jets are obviously counter-acting on the bottom dilution jet. The penetration of the top dilution jet is assisted by the secondary recirculation mentioned above. Because the air supply on the outer liner is in the opposite direction to the combustor outflow and the sloping of the top liner, the top jet has the addition of the dynamic pressure head to its inlet velocity. However, the incoming trajectory is opposed by the primary flow along the top liner and no benefit results. The bottom dilution hole, on the contrary, has the supply air co-flowing in the same direction as the combustor outflow air, therefore does not have the benefit of a dynamic pressure head to increase the inlet jet velocity. At the exit, the flow has a higher horizontal velocity than in the Top-Jet Plane. Immediately downstream of the bottom dilution jet is the vortex that is formed by the crossflow of the jets and the main flow. Similar flow phenomenon is not discovered downstream of the top-dilution jet near the top-liner because the converging geometry suppresses such formation.
Figure 13 shows the measured velocity vectors in Section 3 (Bottom-Jet Plane). The shape of the primary zone recirculation is similar to that of the Inter-Jet Plane. The center of the recirculation is located at (3.302 cm, 1.524 cm) from the reference point. The core of the primary vortex has migrated further downstream due to further decrease in the influence of the primary jet in this section. The maximum velocity measured in the region where the primary jet locates is 15.6 m/s at an angle of 51° above the horizontal plane. The secondary recirculation zone immediately downstream of the primary jet rotates in an elliptic shape with its major axis lying 25° above the horizontal plane, which is much closer to the bottom liner than in the other two sections. The bottom dilution jet is clearly seen in this plane. It attains a trajectory angle of 85° above the horizontal plane and has a maximum measured velocity of 50.8 m/s. Similar to the top dilution jet, the bottom dilution jet penetrates across the flowfield and strikes the top wall at a velocity of about 10 m/s. The flow exiting near the top has higher velocities and angles than at the exit of the other two planes, since the primary flow is co-flowing with the bottom dilution jet flow next to the top liner in the dilution zone.

5.3.2 Total Velocity $V_t$

Section 1 (Top-Jet Plane)

The contour plot and the surface plot of the total velocity $V_t$ are shown in Figs. 14 and 15. The low velocity magnitude at the central part of the primary zone is seen as a low plateau surrounded by a layer of higher velocity flow along the liner wall which is formed by flows from the wall louvres. The primary jet at the bottom is seen with a double peak. This may be due to the relatively large spacing between data points in this high velocity gradient region (the measurements could not resolve the velocity profile of the primary jet). The maximum total velocity in the entire flowfield is found in the top dilution jet with the centerline of the jet looking like a ridge in the surface plot. On either side of it, the velocity magnitude decreases rapidly, thus resulting in high velocity gradient around this dilution jet. Due to the discrete data points measured along the liner wall, the surface at the boundary is not drawn smoothly.

Section 2 (Inter-Jets Plane)

Figures 16 and 17 show the contour and surface plots of the total velocity $V_t$. The primary zone flow structure here is similar to that of Section 1. There seems to be no well defined vortex center point, but rather an elongated strip as the center of the primary vortex, as shown in the contour plot of Fig. 16. This elongated center has velocities below 1 m/s and its formation can be explained as follows. Because the primary jet has a minor influence on the on-coming flow upstream of it in this section, lesser amount of the primary flow is found recirculating back into the primary vortex, thus resulting in a decrease in the flow velocity about the vortex core. The top louvres provide high velocity flow along the liner and shear
or drag) along the slow returning primary flow under it. Therefore, the vortical flow extends further upstream and downstream in the longitudinal direction. The spread of the primary jet is clearly seen in this section, except that the velocity magnitudes have decreased and the depth of penetration across the flowfield is reduced. In the dilution zone, the expansions of the top and bottom dilution jets are seen having an influence on the flowfield in this plane. Although both dilution jets have depth of penetration across the flowfield, the top jet has a lower decay rate than the bottom jet. This is not surprising because both the primary flow and the top dilution jet flows are crossing with the bottom dilution jet flow and are overwhelming it. Another interesting feature here is that the top and bottom dilution jets run side by side across the flowfield in opposite directions as seen in Fig. 17 with the top jet on the upstream side. At the exit plane, the velocity magnitude is higher here than in Section 1, which indicates that more mass leaves through this plane than in the Top-Jet Plane as substantiated by the \( \bar{u} \) data below. This is foreseeable since the primary, top and bottom dilution jets are not all positioned in this section and their influences are less; hence more mass will go around those strong jets and will leave via the in-between sections.

Section 3 (Bottom-Jet Plane)

The contour and surface plots of the total velocity \( V_t \) are shown in Figs. 18 and 19. The layer of flow along the liner in the primary zone has slightly lower \( V_t \) than in Section 2. The gradient of \( V_t \) towards the center is also lower here. Since the top dilution jets are quite distant from this section, the primary vortical flow extends further downstream. However, \( V_t \) of the primary jet has increased slightly due to the expansion of the next primary jet which is only 2.08 mm away. The maximum value of \( V_t \) in the primary jet remains the same as in Section 2, except that the location is shifted further downstream. The secondary recirculation has also a decrease in \( V_t \). In the dilution zone, the bottom dilution jet is seen ejecting from the bottom hole across the flowfield towards the top liner. Its feature is very similar to the top dilution jet as seen in Section 1. At the exit plane, \( V_t \) is the highest amongst all three sections investigated. This is due to the flow from the primary zone entering the dilution zone mainly along the upper part of the combustor and the absence of the top dilution jet here. Together with the crossflowing bottom dilution jet that cuts across the flowfield and impinges on the top liner, \( V_t \) is the highest amongst all three measured sections.

5.3.3 Velocity \( \bar{u} \)

Section 1 (Top-Jet Plane)

The contour plot and the surface plot of the longitudinal velocity \( \bar{u} \) are shown in Figs. 20 and 21. Along the top liner in the primary zone, the \( \bar{u} \) velocity is negative. Therefore it is drawn below the zero reference plane like a valley in the surface plot. On the other hand, \( \bar{u} \) is positive along the bottom liner and is thus drawn above the zero level. The primary jet is seen ejecting from the bottom liner, cutting across the flowfield and reaching the top liner. The maximum \( \bar{u} \) in the primary jet is 13 m/s near the inlet. Immediately downstream of the primary jet is
the secondary zone with \( \bar{u} \) being negative near the bottom liner as the flow reverses its flow direction. Since the top dilution jet has an orientation toward the primary zone, \( \bar{u} \) is negative in this jet. Because the strength of the primary flow engaged in the crossflow in the dilution zone is weaker than the dilution jet, the primary flow goes around the jet and does not prevent the penetration of the jet in the combustor. From the area where the top dilution jet impinges on the bottom liner, \( \bar{u} \) has increased rapidly at the bottom liner due to the turning of the jet downstream after the splashing. A very low velocity zone is seen in the wake area of the bottom dilution jet. The local minimum is zero. No recirculation is found downstream of the top dilution jet and \( \bar{u} \) becomes positive across the exit plane.

**Section 2 (Inter-Jets Plane)**

The contour and the surface plots of \( \bar{u} \) are shown in Figs. 22 and 23. In the primary vortical flow region, the longitudinal velocity \( \bar{u} \) and its gradients in the transverse direction is higher here than in Section 1. The \( \bar{u} \) momentum of the primary jet has decreased and the peak has dropped to 12.7 m/s, and is located further downstream from the jet inlet. A larger secondary recirculation zone with negative \( \bar{u} \) can be seen downstream of the primary jet. The influence of the dilution jets is not obvious here compared with the other two sections. At the exit plane, \( \bar{U} \) is substantially higher than in Section 1. Downstream of the bottom dilution jet flow near the bottom liner, the low velocity wake flow is also seen with velocity of about 3 m/s, which is higher than that in the last section.

**Section 3 (Bottom-Jet Plane)**

Figures 24 and 25 show the contour and surface plots of \( \bar{u} \). In the primary zone, the profile of \( \bar{u} \) is similar to that in Section 2. The influence of the primary jet is also seen diminished and the local maximum of \( \bar{u} \) is 11.6 m/s in the primary jet flow, which is almost identical to that in Section 2 except that the location has moved further downstream of the jet. The smaller negative \( \bar{u} \) area downstream of the primary jet indicates that the secondary recirculation zone is lying low and hugging the bottom liner. In the dilution zone, the bottom dilution jet cuts across the flowfield with its orientation leaning towards the downstream side as shown by the \( \bar{u} \) contours. A local maximum of 12.9 m/s is seen on the downstream of the jet. Along the top liner near the exit plane, the velocity \( \bar{u} \) is in the neighborhood of 10 m/s. At the exit plane, \( \bar{u} \) is higher than in the previous two sections. In the wake of the bottom dilution jet the magnitude of \( \bar{u} \) is about 3 m/s, which is similar to that in Section 2. Hence, more flow leaves the exit plane at the upper portion of the combustor in this section than in the other two sections.
5.3.4 Velocity $\vec{v}$

Section 1 (Top-Jet Plane)

Figures 26 and 27 show the contour plot and surface plot of the transverse velocity $\vec{v}$. Since $\vec{v}$ is generally small and negative due to the flow direction of the vortex in the frontal portion of the primary zone, the surface of $\vec{v}$ is seen as a flat plane slightly slanted downward at the front dome and below the zero level. The primary jet causes entrainment around it and $\vec{v}$ is seen positive on either side of this jet. The maximum $\vec{v}$ above the primary jet inlet is 35 m/s. Unlike U, the velocity $\vec{v}$ coming from the primary jet decreases in magnitude rapidly and it is almost insignificant halfway as it cuts across the flowfield. In the secondary recirculation zone, the magnitude of $\vec{v}$ decreases from the primary jet to the center of the recirculation and then increases into the dilution zone with the sign being negative. The top dilution jet is seen as having a very strong negative $\vec{v}$ velocity with very sharp velocity gradient on either side of it. All the way from the top liner to the bottom liner, $\vec{v}$ is continuously negative which indicates that the top jet impinges on the bottom liner instead of turning the flow direction before reaching the bottom liner. The exiting flow is seen in the velocity vector plot as almost horizontal, thus $\vec{v}$ is at about zero level along the exit plane.

Section 2 (Inter-Jets Plane)

Figures 28 and 29 are the contour and surface plots of the velocity $\vec{v}$. The velocity and its gradient in the longitudinal direction is found higher in this section than in the previous one. Note that there is almost no $\vec{v}$ velocity gradient in the transverse direction near the front dome of the combustor as indicated by the near vertical contour lines in Fig. 28. Velocity $\vec{v}$ has a wide near-zero area at the elongated vortex center. The maximum of $\vec{v}$ and its gradient of the primary jet is diminished. The maximum value here is only 9 m/s compared with 35 m/s in Section 1. Below the double bend top louvres, a small negative V zone is found. This possibly comes from the thumb-naill louvre located between the double bend top louvres, whose flow is oriented in the negative $y$ or tangential direction towards Section 3. In the dilution zone, the top jet has spread into this section, but the magnitude of $\vec{v}$ has decreased. The local minimum of $\vec{v}$ coming from the top jet is only $-16.8$ m/s instead of $-45.5$ m/s as seen in the last section. The location of this local minimum is much further downstream of the jet inlet. Besides a drop in the velocity magnitude, the velocity gradient is also found to be reduced here. The bottom jet is seen clearly just downstream of the top jet decaying rapidly as it penetrates across the flow. The maximum value of $\vec{v}$ near the bottom dilution jet inlet is 17.2 m/s. An explanation of the lack of penetration of the bottom dilution jet in this section has been given above in the discussion of the total velocity $V_t$. At the exit, the flow is not leaving as horizontally as in Section 1.

Section 3 (Bottom-Jet Plane)

Figures 30 and 31 show the contour and surface of $\vec{v}$. In the primary zone, the negative $\vec{v}$ region has expanded further downstream because no top dilution jet hinders the primary vortex from
extending downstream. However, the influence of the next primary jet is seen here with \( \overline{\nu} \) and its gradient is slightly higher than in Section 2. The peak value of the primary jet is now 12.3 m/s compared with 9.0 m/s in Section 2. The secondary recirculation is also seen stronger downstream of the primary jet. The sudden expansion of the air flow from the outer liner into the bottom dilution hole created very high \( \overline{\nu} \) and gradient of \( \overline{\nu} \) at the bottom dilution jet especially on the upstream side of the jet. The maximum \( \overline{\nu} \) is 50.4 m/s near the jet inlet. As the dilution jet penetrates across the flowfield, \( \overline{\nu} \) decays monotonously to about 2 m/s near the top liner. \( \overline{\nu} \) is small here because the primary flow enters and leaves the dilution zone along the top liner nearly parallel to the liner wall and its momentum is high enough to turn the bottom dilution jet flow towards downstream. At the exit plane, \( \overline{\nu} \) is entirely positive, except in the wake region where the jet expands and turn downstream.

5.4 Reynolds Stresses \( \overline{u^2}, \overline{v^2} \) and \( \overline{uv} \)

With the 2-D LDV system, one set of the beams will be blocked by the combustor wall liner when measurements are made close to the wall. The set of beams that is not blocked will obtain one measurement on-axis and another off-axis to obtain \( \overline{u} \) and \( \overline{v} \). Therefore, values of \( \overline{u^2} \) are obtained very close to the top and bottom liners, and \( \overline{v^2} \) are obtained very close to the front dome.

5.4.1 Normal Stress \( \overline{u^2} \)

Section 1 (Top-Jet Plane)

The measured normal stress \( \overline{u^2} \) along the longitudinal axis is shown on the contour and surface plots in Figs. 32 and 33. In the primary zone, \( \overline{u^2} \) is below 10 m²/s² in the primary vortex with its magnitude increasing from the bottom liner towards the top liner. Then it decreases slightly to 6 m²/s² close to the top liner. On either side of the primary jet, \( \overline{u^2} \) increases significantly with higher gradient along the upstream shear layer. The local maximum is 121 m²/s² further downstream from the primary jet inlet. In the secondary recirculation zone, \( \overline{u^2} \) varies from 20 – 40 m²/s². In the dilution zone, high gradient exists on the upstream of the top dilution jet where the flow from the primary zone crosses with the dilution jet. The maximum here is 175.5 m²/s² on the upstream of the jet. However, no symmetrical profile along the centerline of the jet (as in the case of a free jet, see for example Sislian, Jiang and Cusworth 1988) is seen here. This is due to the fact that the jet flow encountered here is actually a flow expansion through a slot hole with the effects of cross flow and the spacing between data points is rather coarse in properly resolving the details of this jet flow. The intensity \( \overline{u^2} \) is seen decaying rapidly as it cuts across the flowfield towards the bottom liner. Downstream of the top dilution jet, \( \overline{u^2} \) decreases to near 20 m²/s² at the top liner and around 30 m²/s² about the centerline. In the wake flow of the bottom dilution jet, \( \overline{u^2} \) drops to 16 m²/s² at the exit plane.
Section 2 (Inter-Jets Plane)

The contours and surface plots of \( \overline{u'^2} \) are depicted in Figs. 34 and 35. The level of the normal stress in the primary vortex is similar to that in Section 1. The magnitude of \( \overline{u'^2} \) increases from a low of \( 2 \) \( m^2/s^2 \) near the bottom of the front dome to a peak of \( 9.6 \) \( m^2/s^2 \) near the top before it decreases slightly towards the top liner. The relatively higher values at the top part of the primary vortex is due to the fact that part of the primary jet flow is drawn into the upper part of the primary vortex. In other words, the large magnitude of fluctuation of \( \bar{u} \) velocity is being convected and diffused into the primary vortex by the mean flow, thus resulting in higher normal stresses in the upper part of the primary vortical flow. Since there is a layer of cooling film along the liner formed by the louvres, which itself issues low \( \overline{u'^2} \) into the flow, \( \overline{u'^2} \) is thus smaller near the top liner. On the upstream of the primary jet, high gradient of \( \overline{u'^2} \) is found. The maximum value in the primary jet is now only \( 75.4 \) \( m^2/s^2 \) compared with \( 121 \) \( m^2/s^2 \) in Section 1, because \( \overline{u'^2} \) of the primary jet decays as the jet expands. This maximum is located at the crossing of the edge of the primary jet and the secondary recirculating flow. Within the secondary recirculation zone, \( \overline{u'^2} \) varies between \( 24 \) to \( 50 \) \( m^2/s^2 \). In the dilution zone, the top jet has a higher gradient upstream than downstream. The maximum \( \overline{u'^2} \) of the top jet is almost only half of that found in Section 1. This local maximum of \( 85.7 \) \( m^2/s^2 \) is located further inside the flowfield and slightly moved downstream due to the crossflow of the top dilution jet with the primary flow since the top dilution jet has lost some of its momentum in this plane. Near the bottom liner in the dilution zone is another relative maximum of \( 106 \) \( m^2/s^2 \), contributed by the bottom dilution jet coming from the Bottom-Jet Plane. This is of similar magnitude as that in the top dilution jet. At the exit plane, a higher magnitude of \( \overline{u'^2} \) is seen in this section than in the last section due to higher \( \bar{u} \) exit velocity and \( x \)-momentum transfer. In Fig. 35, the primary jet and the secondary recirculation zone are not shown quite well because irregular data points are given near the bottom liner. However, a comparison of this figure with Fig. 33 shows that the magnitude of \( \overline{u'^2} \) for both the primary jet and the top dilution jet have decreased. The influence of the bottom dilution jet and the higher values at the exit plane are also seen in Fig. 35.

Section 3 (Bottom-Jet Plane)

The contour and surface plots of the normal stress \( \overline{u'^2} \) are shown in Figs. 36 and 37. The magnitudes of \( \overline{u'^2} \) in the primary zone are very similar to the ones in Section 2 with a local minimum of \( 1.7 \) upstream of the primary jet. Near the bottom liner, \( \overline{u'^2} \) increases from this minimum location to \( \sim 4 \) \( m^2/s^2 \) towards the bottom liner and to \( \sim 10 \) \( m^2/s^2 \) near the top liner. Along the shear layer of the primary jet, \( \overline{u'^2} \) has high gradient and reaches a maximum of \( 80.4 \) \( m^2/s^2 \) at about the centerline of the primary jet close to the jet inlet. This maximum is slightly higher than the one in Section 2. The secondary recirculation zone has a magnitude of \( \overline{u'^2} \) in the vicinity of \( 40 \) \( m^2/s^2 \). In the dilution zone near the top liner, a relative minimum of \( 10.9 \) \( m^2/s^2 \) is found where the primary flow enters the dilution zone and before the bottom dilution jet influences this primary outflow. High gradient of \( \overline{u'^2} \) is seen along the upstream shear layer of the bottom dilution jet. A peak of \( 83.3 \) \( m^2/s^2 \) is seen near the jet inlet and a higher peak of \( 101 \) \( m^2/s^2 \) can be seen further downstream of the jet. This maximum is almost
the same as that in Section 2, except the location is moved downstream. At the exit plane, $\overline{v'^2}$ is also very similar to those in Section 2 with $25 \text{ m}^2/\text{s}^2$ near the top liner, $\sim 40 \text{ m}^2/\text{s}^2$ near the centerline and $\sim 25 \text{ m}^2/\text{s}^2$ in the wake flow above the bottom liner.

5.4.2 Normal Stress $\overline{v'^2}$

**Section 1 (Top-Jet Plane)**

The contour and surface plots of the measured normal stress $\overline{v'^2}$ along the transverse axis are shown in Figs. 38 and 39. In the primary vortex, $\overline{v'^2}$ increases from $2 \text{ m}^2/\text{s}^2$ near the front dome to a local maximum of $13.8 \text{ m}^2/\text{s}^2$ about the central region of the vortex. High gradients of $\overline{v'^2}$ are seen on both sides of the primary jet. The highest value measured in the center of the jet is $183.4 \text{ m}^2/\text{s}^2$. In the secondary zone, $\overline{v'^2}$ decreases to a measured low of $\sim 11 \text{ m}^2/\text{s}^2$. An asymmetrical profile of $\overline{v'^2}$ in the top dilution jet is seen clearly in Fig. 39 issued from the jet. The profile of low $\overline{v'^2}$ along the central axis of the jet and peak values at the shear layers on both sides of the jet are typical of a free jet. The peaks are $302.3$ and $326.4 \text{ m}^2/\text{s}^2$ upstream and downstream of the dilution jet, respectively. The peak is slightly lower on the upstream side due to the crossflow with the on-coming primary flow. The stress $\overline{v'^2}$ is seen decaying as the jet cuts across the flowfield. At the exit, $\overline{v'^2}$ varies from about $60 \text{ m}^2/\text{s}^2$ near the top liner to about $22 \text{ m}^2/\text{s}^2$ in the wake flow of the bottom dilution jet near the bottom liner.

**Section 2 (Inter-Jets Plane)**

The contour and surface plots of $\overline{v'^2}$ are depicted in Figs. 40 and 41. There is a lower degree of fluctuation in velocity $\overline{v}$ in the primary vortex of this section than in Section 1. This is because the strength of the primary jet has decreased in this section, and less mass from the primary jet is being recirculated back into the primary zone. Consequently, lower $\overline{v'^2}$ is found in the primary zone. As shown in Fig. 40, the magnitude of $\overline{v'^2}$ in the primary vortex is below $5 \text{ m}^2/\text{s}^2$ in comparison to $13.8 \text{ m}^2/\text{s}^2$ of the previous section. The gradient of $\overline{v'^2}$ along the shear layer of the primary jet is also lower. The peak value measured in the primary jet flow is $59.6 \text{ m}^2/\text{s}^2$, which is only $32\%$ of the peak value in the primary jet of Section 1. The minimum value of $\overline{v'^2}$ inside the secondary recirculation zone is $7.5 \text{ m}^2/\text{s}^2$ compared with $\sim 11 \text{ m}^2/\text{s}^2$ in the previous section. In the dilution zone, there is a peak of value $402.2 \text{ m}^2/\text{s}^2$ near the top liner due mainly to the top dilution jet flow and another peak of value $610.7 \text{ m}^2/\text{s}^2$ near the bottom liner due mainly to the bottom dilution jet flow. The peak at the top has a higher value than that in Section 1 because of the echoing fluctuation between the top and bottom dilution jets in this plane. The high gradient on both sides of the dilution jets are quite well portrayed by the surface plot in Fig. 41. Downstream of these opposing dilution jets, $\overline{v'^2}$ decays continuously to about $60 \text{ m}^2/\text{s}^2$ near the top and $26 \text{ m}^2/\text{s}^2$ near the bottom in the exit plane.
Section 3 (Bottom-Jet Plane)

Depicted in Figs. 42 and 43 are the contour and surface plots of normal stress $\overline{v'^2}$. In the primary zone, $\overline{v'^2}$ is very small with a magnitude varying from $2 \text{ m}^2/\text{s}^2$ near the bottom liner and the front dome to $6 \text{ m}^2/\text{s}^2$ near the top liner. The magnitude of $\overline{v'^2}$ found in this primary zone is just as low as that in Section 2. On the shear layer upstream of the primary jet, $\overline{v'^2}$ is higher here than in Section 2. Even though this section is further away from the primary jet inlet in Section 1, the maximum of $48.9 \text{ m}^2/\text{s}^2$ at the primary jet flow is of the same magnitude as the one found in Section 2. This may be due to the fact that this section is close to the next primary jet inlet. A local maximum of $35.3 \text{ m}^2/\text{s}^2$ is seen further downstream of the primary jet flow where this jet is being compressed by the primary recirculation above and the secondary recirculation zone below it. Just slightly downstream of this local maximum, the primary flow splits up, with part of the flow being drawn into the secondary recirculation and the other part being drawn by the bottom dilution jet towards the top liner and leaving along the top liner. A minimum value of $10.5 \text{ m}^2/\text{s}^2$ is seen just upstream of the dilution jet about where the primary flow splits. At the bottom dilution jet, a peak value of $422.1 \text{ m}^2/\text{s}^2$ is seen near the jet inlet. The gradient of $\overline{v'^2}$ on the downstream of the dilution jet is more gradual than on the upstream. A saddle structure of the $\overline{v'^2}$ profile is also seen in the bottom dilution jet. The stress $\overline{v'^2}$ decreases to about $53 \text{ m}^2/\text{s}^2$ near the top liner and $27 \text{ m}^2/\text{s}^2$ near the bottom liner at the exit plane.

5.4.3 Shear Stress $\overline{u'v'}$

Section 1 (Top-Jet Plane)

The shear stress profile is shown in the contour and surface plots of Figs. 44 and 45. In the primary vortex, $\overline{u'v'}$ is in general near zero along the front dome and the bottom liner, and increases to $3.8 \text{ m}^2/\text{s}^2$ near the top liner. This indicates that more momentum transfer occurs in the top part of the primary vortex than in the lower part, since momentum flux can be conceived as a stress. However, the magnitude of shear stress in the primary vortex is only 2.3% of the maximum found in this section. Because turbulent motion commonly obtains its energy from shear in the mean flow, it is expected that the turbulence kinetic energy will be small in the lower part of the primary vortex and will increase towards the top liner. High gradients in the shear layer of the primary jet are seen upstream, the measured maximum value $55.1 \text{ m}^2/\text{s}^2$ being near the jet inlet. Downstream of the primary jet, $\overline{u'v'}$ decreases to a low of $0.3 \text{ m}^2/\text{s}^2$ in the secondary recirculation zone. Another high gradient region is seen upstream of the top dilution jet where the primary flow crosses with the dilution jet near the top liner. A maximum of $162.5 \text{ m}^2/\text{s}^2$ is measured upstream of the dilution jet inlet where the crossflow is the strongest. The shear stress then decreases towards the bottom liner as the jet penetrates across the flowfield as shown in Fig. 45. In the wake of the top dilution jet, shear stress decreases. A negative shear stress region is seen below the centerline in the dilution zone. This is created by the interaction of the bottom dilution jet with the crossflow and the top dilution jet. The minimum $\overline{u'v'}$ is $-10.4 \text{ m}^2/\text{s}^2$ in the wake near the bottom liner. Across the
entire flowfield, $\overline{u'v'}$ has the correct sign corresponding to the eddy diffusivity approximation as postulated by Boussinesq.

Section 2 (Inter-Jets Plane)

The shear stress $\overline{u'v'}$ is shown on the contour and surface plots in Figs. 46 and 47. The magnitude of $\overline{u'v'}$ in the primary vortex has decreased slightly compared with Section 1 since the influences from the primary jet and the top dilution jet have reduced in this plane. The gradient on both sides of the primary jet has dropped quite significantly as shown in Fig. 47. The peak value in the primary jet is only 9.4 m$^2$/s$^2$ instead of 48.9 m$^2$/s$^2$ in Section 1. However, an increase to 13.8 m$^2$/s$^2$ in $\overline{u'v'}$ is observed under the top liner double bend louvres. This slight increase is likely due to the thumb-nail louvre injecting flow in the tangential or y direction. Also, the primary vortical flow ends in this location and changes its flow direction turning back to the primary zone. In the secondary recirculation zone, a low of $-1.5$ m$^2$/s$^2$ is seen. In the dilution zone, the shear stress due to the top dilution jet and the bottom dilution jet can be identified clearly in the surface plot. The peak value of $\overline{u'v'}$ in the top dilution jet has decreased to 65.9 m$^2$/s$^2$ from 162.5 m$^2$/s$^2$ in Section 1. The location of this peak has moved from the upstream of the top dilution jet to almost below the top jet inlet. The bottom dilution jet has a peak downstream of the top dilution jet near the bottom liner at 41.1 m$^2$/s$^2$. Then downstream of this bottom dilution jet, a negative shear stress zone is formed by the wake flow. The minimum value of $\overline{u'v'}$ in the wake is $-10.6$ m$^2$/s$^2$, which is nearly identical to the one in Section 1. Over the entire flowfield, $\overline{u'v'}$ agrees with the sign of the Boussinesq hypothesis of turbulence stresses being proportional to the velocity gradients.

Section 3 (Bottom-Jet Plane)

The contour and surface plots of the shear stress $\overline{u'v'}$ are depicted in Figs. 48 and 49. The contour of $\overline{u'v'}$ in the primary zone of this section is very much similar to those seen in the previous sections. The range of shear stress in the primary vortex is from slightly below zero near the front dome to about 3 m$^2$/s$^2$ near the top liner. A local minimum point is seen about the centerline in the primary zone of zero magnitude. The shear layer of the primary jet have high shear stress and gradient near the inlet, reaching a local maximum of 14.9 m$^2$/s$^2$, which is higher than the local maximum value in the primary zone in Section 2. This is because the next primary jet inlet is close to this section. Hence some influences from the next primary jet are experienced here. As shown in the contour plot of the normal stress $\overline{v'^2}$ in Fig. 42, a second maximum of the primary jet is seen further downstream in the core of this jet flow, having a magnitude of 10.8 m$^2$/s$^2$. In the secondary recirculation zone, $\overline{u'v'}$ decreases to $-1.3$ m$^2$/s$^2$. The magnitude of $\overline{u'v'}$ here is less than that seen in Section 1 as expected due to the primary jet inlet location. In the dilution zone, the drastic change in $\overline{u'v'}$ upstream of the bottom dilution jet where the secondary recirculation is being entrained into the dilution jet is seen clearly in Fig. 48. The behavior of $\overline{u'v'}$ of the bottom dilution jet found here stunningly resembles that of a free jet where a minimum and a maximum is situated at the shear layers on either side of the jet axis, and $\overline{u'v'}$ is about zero along the center of the jet as shown in Fig.
48 and 49. Near this dilution jet, $u'v'$ on the upstream and downstream is -61.1 and 78 m$^2$/s$^2$, respectively. These shear magnitudes are only about half of those seen near the inlet of the top dilution jet. This can be explained by the higher momentum transported by the primary jet in Section 1 and subsequently stronger crossflow with the high momentum top dilution jet. Whereas in this section, the crossflow of the high momentum bottom dilution jet is just with the weak secondary recirculating flow. At the exit plane, the shear stress is about 11 m$^2$/s$^2$ in the top half of the plane and decreases to near -6 m$^2$/s$^2$ in the wake of the bottom dilution jet. The shear stress across the flowfield agrees with the velocity gradient hypothesis of the Reynolds shear stress as postulated by Boussinesq.

5.4.4 Shear Stress Intensity SI

Since the complex combustor flowfield is dominated by various flow characteristics, the shear stress intensity SI is defined most appropriately by $\sqrt{|u'v'|}/V_t$. This will provide information on momentum transfer due to turbulence fluctuation relative to the momentum transfer of the mean flow.

Section 1 (Top-Jet Plane)

Depicted in Fig. 50 is the shear stress intensity contour plot of Section 1. Near the base of the front dome in the primary zone, the shear stress intensity is low at 0.1. Since higher shear stress is encountered near the center of the primary vortex, and because the total velocity magnitude decreases towards the vortex center area, the intensity is increased near the vortex center, reaching a local maximum of 7.5 at a point where the total velocity is a local minimum. This shear intensity maximum is fairly high due to $V_t$ being approximately zero. The turbulence intensity and the shear stress intensity are much higher toward the primary vortex center not merely because of the small local mean velocity magnitude, but also due to the increase in turbulence. Because this primary vortex is turbulent and the flow is truly three-dimensional, there are high levels of fluctuating vorticity due to vortex stretching. The primary jet is seen to have fairly constant shear stress intensity between 0.2 and 0.3 along its flow trajectory. In the secondary recirculation zone, the intensity has increased but not to the extent found in the primary vortex. Upstream of the top dilution jet is the shear layer where the shear stress intensity contours run along the direction of the dilution jet. The long strip of contour line here has an intensity $\sim$0.7. The location of this strip coincides with the outer edge of the dilution jet as seen in the velocity vector plot in Fig. 11. A local maximum of 1.18 is found where the primary flow crosses with the high momentum top dilution jet flow at a near normal intersection angle. In the core of the top dilution jet, the shear intensity has decreased to about 0.2 with a local minimum of 0.09 below the centerline. Near the bottom liner before the exit plane, the shear stress intensity increases rapidly to unity due to the bottom dilution jet flow. Downstream of the top dilution jet, the outflow is near horizontal and the intensity is near constant at 0.3.
Section 2 (Inter-Jets Plane)

Depicted in Fig. 51 is the shear stress intensity contour in Section 2. Low shear stress intensity can be seen at the lower part of the primary vortex. However, near the elongated vortex center area, the flow is coming in and out of this plane and experiencing a velocity gradient field away from the vortex, thus resulting in higher shear stress intensity. The primary jet flow has a shear stress intensity of about 0.2, which is similar to that seen in Section 1. The secondary recirculation zone has a slightly higher intensity of 0.6. At the end of the primary vortex just beneath the thumb-nail louvre, the tangential flow from the thumb-nail louvre is influencing the primary flow, as shown in Fig. 12, creating higher shear stress. However, the total flow velocity magnitude has decreased as the flow changes its flow direction and thus resulting in the shear stress intensity being higher than the primary jet flow below it. Upstream of the top dilution jet and at the edge of the shear layer, the local maximum of the intensity is 0.92 where the on-coming primary flow crosses the top dilution jet at a near normal intersection angle. The core of the top dilution jet flow that can be seen from the top liner right down to the bottom liner in Fig. 17 has a smaller shear stress intensity of 0.4. Below the centerline, a local maximum of 0.89 occurs at a point where the negative transverse momentum of the top dilution jet has been canceled by the positive transverse momentum of the bottom dilution jet, resulting in a purely longitudinal momentum in this part of the flow. In the wake flow of the bottom jet, the intensity increases slightly to 0.6. Since the flow is leaving near horizontal and uniform at the exit plane, the shear stress intensity is quite constant over a large region at about 0.3.

Section 3 (Bottom-Jet Plane)

Figure 52 shows the shear stress intensity contour plot of Section 3. Near the front dome liner and the lower part of the primary vortex, the intensity is quite low at about 0.1 because the shear stress is low, even though the velocity near the liner is high. The intensity then increases towards the primary vortex core and two local maxima of magnitude of about 14 are seen in the vortex center area. Three factors may contribute to such high shear stress intensity here. First, the shear intensities in the core area are high due to very small velocity magnitude. Second, the accuracy of the velocity measurement at such low magnitude is crucial. An error of 1 m/s can easily alter the intensity more than ten times. Third, there is a crossflow between planes, especially at the vortex core. Thus a w velocity component exists but is not accounted for in the calculation of the shear stress intensity. The intensity of the primary jet is quite low, about 0.2. The end of the primary vortex underneath the top double-bend louvres has higher shear intensity due to low velocity magnitude and unaccounted flow between planes and flow from the thumb-nail louvre. There is also an increase in the intensity in the secondary recirculation zone. A local minimum of 0.04 is seen in the primary flow as it enters the dilution zone before it encounters the dilution jet. A maximum of 0.93 is located right where the primary jet flow has to change its flow direction swiftly upwards under the strong influence of the bottom dilution jet. Higher shear stress intensity gradient is seen upstream of this bottom dilution jet because the flow of the secondary recirculation is opposing the flow trajectory of the dilution jet. In the dilution jet, the shear stress intensity is as low as 0.1. The intensity is then increased
slightly to 0.4 downstream in the wake of the jet. Near the top liner and the exit plane, the exiting flow has rather constant shear stress intensity of 0.2 over a vast region of the dilution zone.

5.4.5 Reynolds Stress Correlation Coefficient $C_{uv}$

A measure for the degree of correlation between two velocity fluctuations $u'$ and $v'$ is defined by the Reynolds stress correlation coefficient as $C_{uv} = \frac{\overline{u'v'}}{\sqrt{\overline{u'^2} \cdot \overline{v'^2}}}$. Hence, if $u'$ and $v'$ are independent of each other, $C_{uv} = 0$. If $C_{uv} = \pm 1$, their correlation is said to be perfect.

In many turbulent shear flows, $|\overline{u'u_j}| \sim 0.4 \left( \overline{u_i^2} \cdot \overline{u_j^2} \right)^{1/2}$, which is in great contrast with the correlation coefficient for molecular motion (of the order of $10^{-6}$).

*Section 1 (Top-Jet Plane)*

The Reynolds stress correlation coefficient contour plot for Section 1 is shown in Fig. 53. Similar to the shear stress contour plot in Fig. 44, $C_{uv}$ is small in the front dome and lower part of the primary zone. Near the top liner in the primary zone, $C_{uv}$ has increased up to 0.4. This increase is due to the fact that the high shear stress along the shear layer of the primary jet is transported by the turbulent eddies which are recirculating back into the primary zone along the upper part of the primary zone. The maximum at the shear layer of the primary jet is 0.51. A decrease in the correlation coefficient is found in the secondary recirculation zone. The peak coefficient in this section is 0.7 which occurs at the outer edge of the upstream shear layer of the the top dilution jet. In the wake flow of the bottom dilution jet, the shear stress has reached a minimum, and so does the correlation coefficient reaching a local minimum of -0.26. The upper part of the exit plane is ~0.1 and fairly even since the outflow is quite uniform in velocity.

*Section 2 (Inter-Jets Plane)*

The correlation coefficient for Section 2 is shown in Fig. 54. $C_{uv}$ varies from about -0.2 in the front dome to about 0.4 near the top liner in the primary zone. The negative correlation coefficient region upstream of the primary jet seen in Section 1 is not evident here since the primary flow is no longer strongly influenced by the primary jet in changing its flow direction and creating high shear stress in this section. Hence these correlation plots confirm the higher shear gradient seen in Section 1. The influence of the thumb-nail louvre near the top wall is seen in the end region of the primary vortex. The peak coefficient in this plane is located here with a value of ~0.6. Upstream of the opposing dilution jet flow, regions of high coefficient are present. However, in the core of the dilution flow, $C_{uv}$ has decreased to below 0.1 and a local minimum of near zero is seen, which is associated with the top dilution jet. The wake flow near the bottom liner has a local minimum coefficient of about -0.2, which is close to what is seen in Section 1. At the upper part of the exit plane, $C_{uv}$ is slightly higher than that in the last section due to the shearing created by the high volume of the exiting flow leaving between jets in this section.
Section 3 (Bottom-Jet Plane)

Figure 55 shows the Reynolds stress correlation coefficient in Section 3. Like the other two sections in the primary zone, a negative region is present in the front dome area and increasingly higher values occur towards the top half of the primary vortex. Upstream of the primary jet flow near the bottom liner, $C_{uv}$ goes negative to a local minimum of $-0.2$. In the primary jet flow near the bottom liner, $C_{uv}$ increases to 0.3 and then remains at about the same level till the flow is at the end region of the primary recirculation where a local maximum of 0.52 is reached. The core area of the secondary recirculation has dropped to a value near zero. Upstream of the bottom dilution jet where the primary jet flow splits about the centerline, $C_{uv}$ has a local maximum value of 0.39. At the bottom dilution jet inlet, $C_{uv}$ is $-0.42$ upstream and 0.55 downstream. In the core flow of this dilution jet above the centerline, the coefficient is fairly constant at about $-0.2$. Similar to Section 2, a positive coefficient area of about 0.2 is at the upper half and a negative area of about $-0.1$ is at the lower half of the exit plane.

5.4.6 Ratio of the Normal Stress Intensities

The ratio of the normal stress intensities $\sqrt{u'^2/v'^2}$ provides an insight into whether turbulence in the complex flowfield in the combustor is isotropic. If it is anisotropic, then the plot can indicate how strong the anisotropy is and its extent in the flowfield.

Section 1 (Top-Jet Plane)

The ratio of the normal stress intensities for Section 1 is presented in Fig. 56. In the primary zone, $u'^2$ increases in magnitude from 2 (m/s)$^2$ near the bottom liner to about 10 (m/s)$^2$ near the top liner, whereas $v'^2$ is lowest in magnitude along the front dome and bottom liner with a value of 2 (m/s)$^2$ and increases in magnitude towards the center area of the primary vortex. Thus in the lower front dome and bottom liner, the ratio is near unity; above it, the ratio is beyond unity, reaching a maximum of 1.5 near the top liner. Towards the middle of the primary vortex, the ratio decreases to about 0.6. Near the primary jet inlet, the ratio increases slightly to 0.6 from 0.4, and then drops back to a local minimum of 0.41. Further downstream of the jet flow, the ratio increases gradually until the end region of the primary vortex where it is constant at about 0.8. In the secondary recirculation, the ratio increases from the primary jet towards the center of the vortex because of the decrease in $v'^2$. In the dilution zone, a low value of 0.4 is seen near the top dilution jet inlet and the ratio increases on both sides of this jet. One can easily understand from this plot that the flowfield is not isotropic. The anisotropicity does not exist solely in the strong jet flow regions, but also in the vortical or recirculating flow regions. The direct implication of this is that the assumption of turbulence being isotropic in the $k-\epsilon$ turbulence model is invalid in such complex combustor flowfield. In fact, not only are the $k$ and $\epsilon$ values expected to be inaccurately predicted, but also the momentum transport and the mean flow velocity.
Section 2 (Inter-Jets Plane)

Depicted in Fig. 57 is the ratio of the normal stress intensities in Section 2. In the lower half of the primary vortex, the ratio is fairly constant and equal to one. On the upper part of the primary zone, the ratio is above unity with a higher value of 1.2 near the top liner. Upstream of the primary jet flow, \( v'^2 \) increases due to the upward displacement of the primary flow by the primary jet, thus resulting in a region of decreasing values to a local minimum of 0.52 near the jet inlet. As in the previous section, the ratio increases from the primary jet flow to the center of the secondary recirculation zone. At the end region of the primary vortex below the thumb-nail louvre, the ratio of near unity is fairly constant over a large area. In the dilution zone, due to the two opposing dilution jets, the core of this dilution jet flow is about 0.5 in the upper half and 0.4 in the lower half of the combustor. Upstream and downstream of the dilution jets, the ratio is seen to increase with higher gradients upstream. At the exit plane, the ratio is increasing from the top liner to about the middle of the exit plane and then stays quite constant at 0.9. Hence \( u'^2 \) and \( v'^2 \) are more equal in magnitude in the exit plane here than those in Section 1.

Section 3 (Bottom-Jet Plane)

Figure 58 shows the ratio of the normal stress intensities. The entire flowfield has a very similar profile of the ratio found in Section 2. The lower part of the primary vortex has near unity ratio. In the upper part of the primary vortex, a gradual increase in ratio to about 1.2 is seen near the top liner. As in Section 2, due to the increase in \( v'^2 \) upstream of the primary jet, a local minimum of 0.55 is found. In the shear layer of the primary jet, the ratio increases well into the secondary recirculation zone with ratios of about 2.1 near the center of the secondary recirculation zone. At the end of the primary flow below the thumb-nail louvre, again, a near unity ratio region is found. Just upstream of the dilution jet, a local maximum of 1.35 can be seen. This occurs where the primary jet flow splits with one part going downstream together with the dilution jet, and the other part being drawn into the secondary recirculation. In the dilution zone, and on the upstream of the bottom dilution jet, the shear layer has a region of low rms ratio of about 0.4. The higher gradient upstream is due to the crossflow of the jet with the primary flow entering the dilution zone and the opposing flow of the bottom dilution jet with the secondary recirculation. At the exit, the rms ratio of unity is seen almost across the entire plane.

5.4.7 Turbulence Kinetic Energy \( k \)

Section 1 (Top-Jet Plane)

The contour and surface plot of \( k \) are shown in Figs. 59 and 60. Turbulence kinetic energy is rather low in the primary zone due to the fact that turbulence is a cascade process that continues to decay in the absence of any flow mechanism that generates high turbulence
and strong instability inside the flowfield. Higher $k$ with a local maximum of 10.4 m$^2$/s$^2$ is seen in the center region of the primary vortex. At the primary jet inlet, $k$ increases drastically from a low of 2.5 m$^2$/s$^2$ upstream to a maximum of 100 m$^2$/s$^2$. Downstream of the primary jet, $k$ decreases to about 22 m$^2$/s$^2$ at the center of the secondary recirculation zone. Further downstream of the primary jet flow just under the top double bend louvres, $k$ also decreases to about 20 m$^2$/s$^2$. This minimum is located where the primary vortex ends where some primary flow continues to flow towards the top liner and enters the dilution jet while some recirculates back into the primary vortex. In the dilution zone, the saddle-like profile of $k$ generated by the top dilution jet is seen clearly in both figures. The peaks of $k$ near the jet inlet are 239 and 188 m$^2$/s$^2$ with a low at the saddle of 133 m$^2$/s$^2$. The gradient of $k$ upstream of the dilution jet near the top liner is higher than that downstream due to the crossflow with the primary flow. However, below the centerline and upstream of the jet, the gradient of $k$ has decreased due to the fact that part of the flow transports and diffuses $k$ into the secondary recirculation zone. In the wake of the top dilution jet, $k$ expands out and decays gradually to about 50 m$^2$/s$^2$ near the exit plane. Below the centerline near the exit, the wake flow of the bottom dilution jet is seen as having a similar level of $k$ as that of the secondary recirculation zone with a magnitude of about 20 m$^2$/s$^2$.

Section 2 (Inter-Jets Plane)

The contour and surface of $k$ are shown in Figs. 61 and 62. Turbulence kinetic energy varies from 2 m$^2$/s$^2$ near the bottom liner to 6 m$^2$/s$^2$ near the top liner in the primary vortex which is smaller than those in Section 1. The gradient of $k$ at the shear layer of the primary jet is more moderate here than in the Top-Jet Plane. The maximum value of $k$ near the jet inlet is $\sim$75 m$^2$/s$^2$ compared with 100 m$^2$/s$^2$ in the previous section. In the secondary recirculation zone, $k$ decreases to about 15 m$^2$/s$^2$ at the center. Further downstream of the primary jet flow above the secondary recirculation zone, $k$ is fairly even over a large area with values between 20 to 30 m$^2$/s$^2$. In the dilution zone, due to the spread of the two opposing dilution jets, the level of turbulence in this section is even much higher than in the other sections. Two peaks are found in this zone — one near the top liner and the second near the bottom liner. The peak at the top has a magnitude of 240 m$^2$/s$^2$ which is almost identical to the peak of the top dilution jet found in Section 1. The bottom peak has a magnitude of 339 m$^2$/s$^2$ which is the highest level in the dilution zone among the three sections investigated. This maximum is the result of the intersection of the top and bottom dilution jets. If both opposing dilution jets had similar strength, then this maximum would not occur at the centerline. As seen clearly in the contour plot of Fig. 61, this is not so the case. The maximum is leaning on the bottom portion of the sector because of the secondary recirculation where the primary flow is added to the top dilution jet flow coming downward in the dilution zone toward the bottom liner and offsetting the location of this maximum. Large values of $k$ in the dilution zone simply dwarf the local maximum of the primary jet in the surface plot of Fig. 62. A comparison of Figs. 60 and 62 depicts clearly the extremely different profiles of $k$ in a plane away from the jet inlet. The gradients of $k$ upstream and downstream of the opposing jet flow are also much higher in this section than in the other two sections. At the exit plane, $k$ is about 40 m$^2$/s$^2$ which is approximately of the same magnitude as in the other sections. These results indicate
that little mixing takes place in the primary vortex. Moderate turbulence mixing occurs at the primary jet and the secondary recirculation zone, and extremely high inter-mixing exists in the dilution zone.

Section 3 (Bottom-Jet Plane)

Depicted in Figs. 63 and 64 are the contour and surface plots of $k$ in the third section. The distribution of $k$ in the primary zone is similar to that seen in Section 2. The magnitude of $k$ below the centerline in the primary vortex is nearly equal in these two sections. Above the centerline, the gradient and magnitude of $k$ is slightly higher than in Section 2. This is because high turbulence levels generated in the primary jet in this section are convected and diffused back into the primary recirculating flow due to a slight lengthening of the vortical flow, thus contributing to the slightly higher $k$ near the top liner. Since this section is further away from the primary jet of Section 1, the magnitude of $k$ contributed by the jet has decreased and a local maximum of only 43.8 m$^2$/s$^2$ is seen. Further downstream along the primary jet flow, $k$ decreases slightly and levels off at about 25 m$^2$/s$^2$ above the secondary recirculation zone. The same magnitude of $k$ is also seen within the secondary recirculation flow. Just upstream of the dilution zone about the centerline, a local minimum of 14 m$^2$/s$^2$ is found. This occurs where the primary jet flow splits, one part moving upward towards the top liner due to the entrainment of the bottom dilution jet, and the other being drawn into the secondary recirculation zone. In the dilution zone, the gradient of $k$ upstream of the bottom dilution is much higher than downstream. A local maximum of 237.2 m$^2$/s$^2$ is seen in this region. This maximum is almost identical to the maximum found upstream of the top dilution jet in Section 1 which is 239 m$^2$/s$^2$. At the exit plane, $k$ has decayed down to about the same magnitude of 40 m$^2$/s$^2$ as in Section 2.

5.4.8 Ratio of $|u'v'|/k$

Section 1 (Top-Jet Plane)

Figure 65 compares the magnitude of the shear stress $u'v'$ with the turbulence kinetic energy $k$ using the ratio of $|u'v'|/k$. In the primary zone, both $u'v'$ and $k$ have smaller values in the lower half of the combustor and higher values toward the top liner. The ratio of $u'v'$ and $k$ also indicated a similar trend here with smaller value near the bottom liner and higher value towards the top liner. Since the shear stress is a measure of momentum transfer by turbulence motion whereas turbulence kinetic energy relates to the amount of turbulent mixing in the flow, the figure shows that near the top liner there is relatively more momentum transfer in the flow due to turbulence than due to the mixing process. The highest ratio near the top is about 0.4. At the primary jet, the ratio is about 0.5 near the jet inlet. The drop-off of this ratio on both sides of the shear layer can be seen clearly in the figure. In the core of the jet, the ratio is about 0.25 until above the centerline where the ratio increases to 0.45 and then decreases towards the top liner. In the secondary recirculation zone, the ratio decreases from about 0.25 at the shear layer to below 0.05 towards the center of the recirculation. Slightly
downstream of the secondary recirculation zone close to the centerline, a local maximum of 0.5 is found where the primary flow has a strong crossflow effect on the top dilution jet. The ratio of $u'v'$ and $k$ has the highest value of 0.75 upstream of the top dilution jet near the jet inlet where the primary flow has to change its flow direction quite suddenly when it approaches the dilution jet. The trajectory of the dilution jet is almost perfectly traced by the contour lines in the dilution zone. In the wake of the top dilution jet, the ratio decreases to near zero.

Section 2 (Inter-Jets Plane)

The ratio of the shear stress $u'v'$ and the turbulence kinetic energy $k$ is shown in Fig. 66. In the primary zone, the ratio is higher near the top than the bottom liner. Referring back to the velocity vector plot in Fig. 17, the velocity magnitude has a smooth and gradual transition from higher values next to the bottom liner to lower values in the vortex center region. Hence lower shear stress and lesser amount of turbulence exist in the lower part of the primary vortical flow. Although higher shear stress and turbulence is found on the upper part of the vortical flow, the rate of increase in $u'v'$ is larger than in $k$. As the primary jet flows toward the top liner, $k$ decays moderately up to about the centerline and then levels off at about 20 $(m/s)^2$, whereas $u'v'$ decreases rapidly initially near the bottom liner and then levels off at about 5 $(m/s)^2$ near the centerline and then increase again (see Fig. 46 and 61). Under the thumb-nail louvre, the primary flow splits up and there is also tangential flow from the thumb-nail louvre, hence high ratio of $|u'v'|/k$ is found in this area with values between 0.5 and 0.6, which is the highest in the entire flow. The secondary recirculation zone has a much smaller value of about 0.1. An almost circular region of high ratio of $\sim 0.4$ is located just upstream of the dilution jets. From the velocity vector and the $u'v'$ plots of Figs. 12 and 46, this region corresponds to the crossing of the secondary recirculating flow with the top dilution jet. Once inside the core of the interaction of the two opposing dilution jets, $k$ dominates the flow, hence reducing the ratio to about 0.1. At the exit plane, $u'v'$ varies from positive in the upper half to negative in the lower half of the combustor, and the zero crossover is near the centerline, thus resulting in a local minimum region about the middle of the exit plane.

Section 3 (Bottom-Jet Plane)

The ratio $|u'v'|/k$ for Section 3 is shown in Fig. 67. Near the front dome of the combustor, $u'v'$ is nearly zero, hence the ratio of $|u'v'|/k$ is about 0.05. A local minimum of near zero is seen in the primary zone just below the centerline where $u'v'$ also is a minimum here. These low ratios imply that shear stress is small and little momentum transfer occurs even though there is still velocity fluctuations. Towards the upper part of the primary vortex, the ratio increases to 0.4 due to higher velocity gradient from the center region of the vortex to the higher velocity flow from the louvres. The ratio increases slightly from 0.05 and levels out at 0.2 in the core of the jet until the centerline where the ratio increases again up to a maximum of 0.515 below the thumb-nail louvre. This maximum is at the peak of the shear stress in the primary flow where the primary vortex ends and the flow changes direction drastically to recirculate back into the primary vortex. On top of this is the flow from the top liner,
possibly coming from the thumb-nail louver, descending and interacting with this returning primary flow. In the secondary recirculation, $u'v'$ is very small and so is this ratio. Just before the dilution zone downstream of the secondary recirculation zone, a local maximum of 0.37 is found where $k$ is a local minimum due to the splitting up of the primary jet flow as mentioned earlier. Above this local maximum is a local minimum of almost zero which is the result of $u'v'$ changing sign from positive to negative here. In the dilution zone, the bottom dilution jet has the ratio ranging from 0.05 to 0.1 in the core flow. Upstream of the bottom dilution jet, the maximum value of $|u'v'|/k$ amongst the three sections is found. The highest ratio of 1.34 located here is a consequence of the extremely high shear stress at the shear layer of the bottom dilution jet which also encounters the secondary recirculation flows in an opposing direction. Downstream in the wake flow of the jet, a local minimum of near zero is seen also due to a sign change in the shear stress. At the exit plane, the ratio is very similar to that found in Section 2 with about 0.1 in the center and lower half, and about 0.2 near the top half of the combustor.

In general, the magnitude of shear stress ranges from 0 to 0.5 times that of the turbulence kinetic energy in most part of the flowfield. The exception comes in the secondary zone upstream of the dilution jets where shear stress has its peaks in the shear layers near the jet inlets.

5.4.9 Turbulence Intensity $TI$

The flowfield in the toroidal vortex combustor is truly complex and no complete experimental data has ever been obtained to provide detailed information on the turbulent flow structure. As described in the previous subsections, it is inappropriate to define the turbulence intensities as in the case of a free jet or wall shear flow with $\sqrt{u'^2}/U$ or $\sqrt{v'^2}/V$ because within the combustor, the various flow features dominate different parts of the flowfield. Therefore $\sqrt{(u'^2 + v'^2)/(U^2 + V^2)}$ is defined as the turbulence intensity here.

Section 1 (Top-Jet Plane)

Figure 68 shows the turbulence intensity in Section 1. Since the primary vortex behaves like a forced solid body rotation with higher velocity on the outer portion of the vortex and the amount of turbulence fluctuations at the front dome and the bottom liner of the primary zone are small, the turbulence intensity is the lowest at the lower front of the combustor with values of about 0.4 to 0.6. Towards the center of the primary vortex, turbulence energy increases whereas the velocity decreases, resulting in higher turbulence intensity. The core of this vortical flow has a turbulence intensity of about 0.8. At the local minimum point of the primary vortex, a peak in the turbulence intensity is formed. This may be quite misleading because turbulence fluctuations at this point are actually very low (see Figs. 32 and 38). Of the primary jet flow, the turbulence intensity is fairly low varying between 0.6 and 1.0. However, the intensity is seen increasing towards the center of the secondary recirculation zone with a maximum of about 0.6. Furthermore, in the dilution zone, a local maximum is located
upstream of the top dilution jet at the shear layer. Near the inlet of the top dilution jet, the turbulence intensity is also a minimum at 0.4. As the jet flows across the flowfield to the bottom liner, the intensity increases only slightly. However, the intensity increases more in the wake flow, and high intensity is found near the bottom liner before the exit plane where the influence of the bottom dilution jet is found.

Section 2 (Inter-Jets Plane)

The turbulence intensity contour plot is shown in Fig. 69. The top-front part of the primary vortex has slightly higher turbulence intensity in this section than in the previous one. The primary vortex has now an elongated center region. Two local maxima with intensity of 9.7 and 16 are found in this center region. The intensity of the latter is close to that found in Section 1. However, the lower part of the primary vortex has lower turbulence intensity than in Section 1. This is due to decreased influence of the primary jet on the primary zone. The turbulence intensity along the primary jet flow has also dropped slightly to about 0.5 compared with 0.6 in the last section. Since more mass leaves the primary zone in this section than in Section 1 and the strength of the primary jet flow has decreased, the secondary recirculation zone lies low and close to the bottom liner. The turbulence intensity within this recirculation is about 2.0, which is similar to that found in Section 1. In the dilution zone, the turbulence intensity is about 1.5 at the top and 2.0 at the bottom. A local peak of 4.7 is seen in the opposing dilution jet flow near the centerline. At the exit plane, the intensity is near unity in the upper half and increased slightly to 2.0 near the bottom liner where the bottom dilution jet wake flow is located.

Section 3 (Bottom-Jet Plane)

The turbulence intensity in Section 3 is shown in Fig. 70. Low turbulence intensity can be seen in the lower part of the primary vortex and increasingly higher intensity towards the vortex center region. Very high turbulence intensity is seen in the center region of the primary vortex. This is not to be interpreted that the maximum turbulence kinetic energy value can be found here. The turbulence kinetic energy is only about $6 \text{ (m/s)}^2$, which is very low compared with the rest of the values in the flowfield (see Fig. 64). The cause of such high intensity is due to the vanishing small velocity magnitude. In the primary jet flow, the turbulence intensity is fairly low at about 0.5 to 0.6 with a local minimum of 0.47 near the centerline. Above this primary jet flow, there are flows recirculating back into the primary vortex, and flows incoming from the thumb-nail louvre and crossflows between planes; the velocity magnitude is thus low and the turbulence intensity high. In the secondary recirculation zone, the intensity increases toward the center where the velocity magnitude decreases. Near the bottom dilution jet inlet, the turbulence intensity in the center part is as low as 0.2. It then increases rapidly upstream at the shear layer with a local maximum of 3.4 situated at the point where the primary jet flow encounters the strong dilution jet and is altering its flow direction to go along with the dilution jet. The velocity magnitude there is a local minimum as seen in Fig. 13. At the upper part of the exit plane, the turbulence intensity is about 0.8 and it increases toward the center of the wake flow of the bottom dilution jet near the bottom liner to about 1.75, where the velocity magnitude is also smaller.
5.5 Combustor Flow Structures in Cold Flow Without Fuel Injection

The descriptions and presentations of the measured flow variables in the last section provided pertinent details of the combustor flowfield. In this section, some of the flow structures inherent to this combustor geometry design and how they are formed will be described based on the present experimental findings.

The flow patterns observed from the LDV measurements substantiated the prominent feature of the well-structured vortex pattern in the primary zone of the PWC toroidal vortex combustor as observed from the flow visualizations of the present sector-combustor using Mie scattering technique and the water tunnel model at PWC, and the CFD results obtained in this study and at PWC. Some combustors employ flameholders, and others use swirlers surrounding the fuel injectors to obtain flame stability. PWC chooses the toroidal vortex approach to recirculate combusted hot gas products back into the primary zone to achieve combustion stability.

The benefits of toroidal vortex structure over flameholders and swirlers are many. There is no protrusion nor exposure of an object in the hottest part of the flowfield, eliminating thus the problem of hot component material failure, and consequently extending engine run time between maintenance. Vortical flow can be easily generated by combustor geometry, locations and ways of introducing airflow into the combustor; therefore circumventing the complexity of swirler devices. Since the flame stability has already been taken care of by the vortex, the fuel injector design can be less complicated and more compact. Also, the fuel injectors can be mounted at will, axially or radially. The fuel injector can also be rotated to take advantage of the primary zone volume in achieving longer residence time of the combustible mixture in the primary zone before the dilution zone. This will also lead to higher angular coverage of each fuel injector which results in fewer fuel injectors, reduced weight, and higher combustion efficiency due to more complete combustion before dilution. However, the low velocity magnitude and low turbulence intensity in the center region of the primary vortex may be of concern because it is likely to provide poor mixing efficiency and subsequently low combustion efficiency and high emission levels. Another concern is that if fresh air films enter the combustor along the liner, it may quit the role of liner cooling and begin reacting near the liner walls. Thus, scheduling of air into various parts of the combustor is crucial in the present toroidal vortex type design.

The streakline plots of the 3 sections are shown in Figs. 71 to 73. The primary zone begins and ends at about the thumb-nail louvre at the top liner. Downstream from the thumb-nail louvre is the dilution or secondary zone. The toroidal vortex in the PW200 combustor is formed by the geometry of the front dome which encourages circular flow in the front-end. By introducing some air to the wall louvres along the liner tangentially, the vortical flow structure establishes itself naturally. If the flow velocity along the liner is increased, the vortex gets stronger. The flow velocity profile generated by the forced rotation velocity distribution with higher velocity along the rim, due to the inlet velocity from the louvres, decreases linearly towards the vortex center. The vortical flow in the center is formed by the shearing action.
between fluid layers. However, little flow recirculation is achieved with only wall louvres as the driving mechanism of the vortex. Also, the vortex will be stretched very far downstream before recirculating back. By introducing a wall jet in the primary zone, the extent of this vortex downstream and the shape of it can be easily controlled by the strength of this primary jet. This jet also enhances the vortex. The air that enters through a wall louvre in the primary zone remains attached to the liner, due to the centrifugal force acting on the flow, until it comes to another wall louvre where it is displaced away from the wall. Consequently, fresh air is available readily along the outer part of the vortex. The core of the vortex then consists mostly of hot combustion product due to recirculation and this region acts as a storage of heat.

The streakline plots of the 3 sections disclose many interesting features of the flowfield. In the Top-Jet Plane (Fig. 71), the displacing of the near wall flow is seen clearly in the primary zone. The primary jet flow forces some of the primary flow to recirculate and this vortical flow is flowing towards the center in which this vortex functions like a sink in this plane; the flow apparently reappears in the neighboring planes which can be considered as a source. In the Inter-Jets Plane (Fig. 72), the vortical flow is seen as a source with flow coming out and moving outward away from the core. Only a small amount of the vortical flow in the core region is being drawn back into the sink seen in Section 1. The tangential flow from the thumb-nail louvre can be seen partially drawn into the primary zone by the vortical flow as well. Further from the primary jet inlet in the Bottom-Jet Plane (Fig. 73), the source-like vortex core has been elongated and the flow from the neighboring plane continues to come out from the vortex center region and leaves the primary zone mostly along the top liner.

Regarding the secondary recirculation, most of the primary jet flow in Section 1 which reaches almost the top liner is seen drawn back down by the top dilution jet and recirculates into the secondary vortex. The secondary vortex behaves as a sink just like the primary vortex. Although the two-dimensional streaklines drawn in a truly three-dimensional flowfield may be misleading in some respect, they do indicate the most possible flow trajectories that the flow would take. However, the entire primary flow being forced back downward by the top dilution jet is plausible. Some of this flow must have taken a trajectory out of this plane to get around the top dilution jet and reach the exit plane. In Section 2, only the primary flow below the centerline is drawn into the secondary recirculation, whereas the portion of the primary flow beyond the centerline finds its way into the dilution zone. However, part of the secondary vortex core acts as a source with flows from the neighboring planes emerging out here, which behaves very much like the primary vortex in this plane. Furthermore, in Section 3, almost all the primary flow leaves the primary zone and none of the primary jet flow is being drawn into the secondary vortex. More of the secondary recirculating flow leaves along the outer edge of the secondary vortex in this section than in the previous due to the assistance of the bottom dilution jet flow.

The local influence of the dilution jets on the flowfield in the dilution zone is strongly three-dimensional. The depth of penetration of the top dilution jet is comparable to that of the bottom dilution jet. The low velocity area below the centerline just before the exit plane is formed by the crossflow of the two opposing dilution jets and the primary flow. Thus these results provide valuable in-depth understanding of the flow pattern inside the present compact
and complex combustor. The slow rotating primary vortical flow is also as three-dimensional as the crossflow in the dilution zone, and the three-dimensional flow in the secondary zone is slightly more subtle amongst all the other flow features.

To understand how the flow patterns can be applied to a reactive flow, assume that fuel is injected into the flowfield from one of the top wall louvres. The fresh air and fuel flowing near the liner will be pre-heated by the hot gases in the core region of the primary vortex and the fuel droplets will begin to vaporize. Concurrently, the fuel will mix with the fresh air. However, the fuel-air mixture will still be too rich for combustion until it encounters the primary jet where high turbulence enhances the mixing process and spontaneous combustion takes place. This jet flow forces some of the hot gases to recirculate back into the primary zone. The other part of the hot combusting flow is displaced by the primary jet towards the top liner where some will cross flow with the dilution jet and some will be drawn into the secondary recirculation. Part of the hot flow in the primary vortex core will migrate to another plane and leave the primary zone close to the top liner as it enters the dilution zone.

Because this hot flow travels towards the top liner, a double-band louvre and a thumbnail louvre are used to cool the top liner. Due to high momentum of the two sets of opposing dilution jets, the primary flow is simply not able to be mixed to a high degree with the dilution air to achieve proper cooling. It can be seen in the Inter-Jets Plane that most of the primary flow can enter and leave the dilution zone via this plane where the influences of the jets have much decreased. From these observations, it may then be beneficial to have the strength of these dilution jets reduced to such magnitude that the primary flow may be able to have more interactions with the jet flows rather than being so overwhelmed that it has to flow around these jets without being effectively cooled before exhausting.
Chapter 6

Combustor Flow With Fuel Injection

Presented and discussed below are the experimental measurements and the reduced data obtained for the cold and hot flows with fuel injection.

6.1 Flow Visualization

Part of the flowfield in the hot flow case with fuel injection could not be visualized by naked eyes with neither optical arrangements nor particle seeding because the combustion of methane in air provided a bluish-white luminous flame. The longitudinal and transverse spread of the flame could be viewed from the side windows of the combustor rig, whereas the transverse and tangential spread of the flame could be viewed from the rear window. The visible flame was video-taped and photographed.

Figure 74 shows the photographs of the flame taken from the side and rear windows of the rig. In Fig. 74(a), the flame is not clearly seen at the fuel nozzle exit probably caused by the acceleration of the flow and the richness of the fuel, since proper mixing for combustion is far from ideal. In the order of an injector exit diameter, the flame is seen with a bright white color in the primary zone. As the fuel cone splashes onto the bottom liner, some fuel flows up towards the front dome louvre, mixes with the primary fresh air and burns. Brighter burning is also seen along the shear layer of the fuel spray cone as seen in Fig. 74(b). At the bottom liner where the spray cone hits the lower part of the front dome, the primary jet is partially entrained and recirculated into the vortex downstream of the fuel jet, hence the flame is seen not spread too far downstream. On either side and behind the fuel injector are the wake flows with very lean fuel-air mixture that cannot sustain combustion. Further away from the fuel injector plane in the tangential direction, the dominance of the fuel jet subsides and the primary vortex becomes the key flow features that provide flame stability. The entrainment and recirculation of the primary jet have reduced and thus the primary flow has higher momentum forcing the primary jet to bend more downstream. The flame at the bottom liner there can be seen with slight increase in its spread downstream. The brightest part of the flame is seen about the centerline further away from the fuel injector where the
primary vortex is fully developed. This brightest combustion flame is seen in Fig. 74(a). The flame is clearly seen contacting the liner at the front dome louvre and the bottom louvre, but it is lifted above the liner as it reaches the primary jet. As most of the primary flow enters the secondary zone near the top liner, the flame is seen spreading further outward in the tangential direction, and comes in contact with the top liners as seen in Fig. 74(b).

### 6.2 Experimental Results

The various sections investigated for the cold and hot flows with fuel injection are shown in Fig. 75. In the sketch, not all the jet inlet holes are shown. In the case of fuel injection, the centerline of the fuel nozzle was in line with the top-jet hole in the center plane, therefore this center plane was used as the reference plane and was referred to as Section 1 (Injector Plane). Only this plane was mapped for the cold flow case. In the hot flow runs, three sections were investigated. The first plane was chosen to be Section 1 again so that comparisons could be made with the cold flow results to determine the effects of combustion on the flowfield. Section 2 was 7 mm away from Section 1 where it was located within the shear layer of the fuel jet. This second plane was carefully chosen so that a better picture and understanding of the behavior of the fuel jet could be obtained. It was believed that the dominance of the fuel jet in the primary zone would vanish in planes that were distant from the injector plane. The criteria in selecting Section 3 were: 1) that it must be far away from the injector where a flowfield similar to those found in the no fuel-injection runs would be located; 2) it must be some distance away from the side window so that no side-wall effects would be encountered; and 3) it must be at the center of one of the bottom dilution holes. Section 3 was finally chosen to be 39.9 mm away from Section 1. Near the flow boundaries, measurements were done at three different sensitivity vector directions with one component of the LDV system to obtain data for the velocity components and stresses. Finer grid increments were used when approaching the liner walls to obtain more data in the shear layers.

### 6.3 Flow Velocities

#### 6.3.1 Velocity Vectors

*Cold Flow Section 1 (Fuel Injector Plane)*

The velocity vector plot of the cold flow with fuel injection in Section 1 is shown in Fig. 76(a). The fuel injector is located directly above the primary jet and upstream of the double-bend louvres. The ignitor is situated in the upper front-end of the sector. In the primary zone, the recirculating flow that dominates the upstream region in the cold flow without fuel injection has now disappeared. In the upper half of the primary zone near the ignitor, a weak vortical flow is found. The air and fuel are injected through the fuel nozzle into the flowfield with a weak swirl to encourage entrainment of the fuel jet with the surrounding air and the
core air. The entrainment can be seen clearly on either side of the fuel jet. The swirling effect is indicated by the small spread of the fuel jet velocity vectors near the fuel nozzle. The trajectory of the fuel jet is about 45°. Since the primary vortex is a low velocity region where the velocity varies from near zero at the center to about 10 m/s near the wall (as seen in the no fuel-injection case), the injected air-fuel mixture easily penetrates right across the primary zone, striking against the bottom liner of the front dome and splashes out. Part of the fuel jet flows upstream toward the louvre at the dome where it further splits into two small recirculation zones — one above and one below the centerline. The elongated recirculating flow above the centerline is created by the upward thrust of the fuel jet and the low pressure region generated by the higher wall layer flow across the ignitor. The lower recirculation zone is the result of the upward splashing of the fuel jet and the incoming fresh air charge from the louvre at the dome. Downstream of the fuel jet near the bottom liner wall is a larger vortex which is set up by the fuel jet’s entrainment effect, the splashing of the fuel jet at the bottom wall, the incoming air flow from the bottom wall louvre, and the primary jet downstream. The primary jet serves three purposes. It keeps this strong vortex from moving downstream because it is this vortical flow that helps stabilize the flame in the primary zone, it enhances the vorticity of this recirculation, and it provides a large volume of fresh air for combustion in the primary zone.

The trajectory angle of the primary jet is approximately 80°, which is much higher than in the no fuel-injection case. Downstream of the primary jet is another low pressure zone where another vortex is found. Comparing with the no fuel-injection case, the vortex here is much larger in size and its center is lying just below the centerline. The high trajectory angle of the primary jet, the flow character below the fuel nozzle, and the size of the vortex downstream of the primary jet suggest that only a small amount of flow from upstream is leaving the primary zone via this plane and the exiting flow would most likely be along the top liner wall. Most of the flow entering the secondary zone originates from the primary jet. In the secondary zone, the top dilution jet is seen entering the flow domain with a trajectory normal to the top liner wall. A small amount of the flow from the primary zone is being entrained into the dilution jet and the rest of the primary flow enters the secondary zone through gaps between dilution jets. Directly below the top dilution hole under the centerline, the velocity magnitude decreases to almost zero and then increases in magnitude but the flow here turns toward the exit plane. This flow character is probably the consequence of the interactions of the crossflow with the opposing dilution jets, and crossflow between planes in the tangential direction. Some portions of the top dilution jet are found entraining into the secondary recirculating flow downstream of the primary jet and some are penetrating right down to the bottom liner. A thin layer of high velocity wall flow above the bottom liner is also seen. The outflow at the exit plane still carries some transverse momentum of the top dilution jet and thus is not leaving as horizontal as in the cold flow without fuel-injection case.

**Hot Flow Section 1 (Fuel Injector Plane)**

The measured velocity vectors of the combusting case in Section 1 is shown in Fig. 76(b). The flow characteristics are almost identical to the cold flow case. Hence much of the discussion above for the cold flow with fuel injection also applies here. The following discussion highlights
the effects of heat addition due to combustion on the entire flowfield investigated. The fuel jet is seen penetrating across the primary zone with much higher momentum, and striking the bottom liner with much higher velocity. The splashing that goes upstream toward the front dome creates a stronger vortex at the inlet of the front dome louvre, and it also causes some flow to recirculate back into the fuel jet on the upstream. The increase in flow velocity along the top liner and this recirculating flow both contribute to the higher vorticity of the primary vortex. The center of this vortex, which is not well defined in the cold flow, is now well formed and clearly depicted in the figure. The center of this primary vortex has moved closer to the top liner. The small recirculation beneath the ignitor has also been pressed closer to the front dome liner and higher above the centerline of the sector. The vortex trapped downstream of the fuel jet is also found much stronger with the center well defined and lying closer to the bottom liner. This vortex is trapped by a triangular shaped flow pattern formed by the fuel jet, the primary jet, and the mixing film of the splashing fuel jet with the bottom louvre air. Since the strength of these three flows have increased due to combustion, the vorticity of this trapped vortex has also increased as a consequence.

Although the primary jet enters the flowfield and flows upward toward the top liner with higher velocity, the flow trajectory toward the fuel injector has not changed. The outflow at the upper half of the sector has gained higher longitudinal momentum but lost almost all its transverse momentum. The recirculation downstream of the primary jet neither changes in size nor moves in space, except that its vorticity has increased without much changes in the magnitudes of the velocities, as indicated by the tighter turning of the velocity vectors. In the secondary zone, the incoming top dilution jet has slightly lower longitudinal momentum in the upstream direction, which is a consequence of the stronger crossflow between the dilution jet and the longitudinal outflow from the primary zone. The vortex, which is created by the inter-mixing action between the top and bottom dilution jets and the crossflow, does not change in space but the velocity has increased. Part of the dilution jet is being entrained into the secondary recirculation and some penetrates as far down as the bottom liner wall. The velocity of the film above the bottom liner wall is also higher. At the exit plane, the spreading top dilution jet is found turning more rapidly and exiting with the mainstream near horizontal about the centerline.

**Hot Flow Section 2**

Measurements of Section 2, which is about one fuel injector exit diameter away from Section 1, is shown in Fig. 77. Many similar flow characteristics as in Section 1 are found here. The fuel jet is not seen entering this plane until just above the centerline of the sector. The recirculation at the upstream of the fuel jet at the bottom of the front dome is better defined. The primary vortex near the ignitor has increased in size with its center moved slightly downstream. The vortex on the downstream of the fuel jet did not change much. Flows from this vortex and the primary jet are being drawn into the fuel jet at the gap between the fuel injector and the appearing fuel jet. The recirculation downstream of the primary jet has decreased in size with the center lowered toward the bottom liner since the center of the primary jet hole is not located in this plane. In the secondary zone, the primary flow enters near the top liner and is being compressed by the strong bottom dilution jet which penetrates
across the flowfield reaching the top liner. The vortex generated by the inter-mixing of the dilution jets and the crossflow is also seen in this plane around the bottom dilution jet.

**Hot Flow Section 3**

However, in Section 3 which is located far away from the fuel injector plane, the entire flow picture changes significantly as shown in Fig. 78. A flowfield very much similar to the cold flow without fuel injection is found in this plane, except that the primary vortex is now elongated vertically. The fuel jet has totally disappeared from the primary zone altogether with the vortices that are generated by its splashing on the bottom liner, leaving behind the toroidal vortex in the front-end of the primary zone. This is the same primary vortex that is located in front of the ignitor in Section 1, except that it has grown in size and its center is lowered toward the centerline. Since this plane is not in line with any primary jet, only some trace of this jet is found. With the massive outflow of the mainstream leaving the primary zone, the secondary recirculation region downstream of the primary jet has decreased drastically in size and it becomes elliptic in shape with the center lying close to the bottom liner. In the secondary zone, the penetration of the bottom dilution jet has diminished due to the high longitudinal flow near the top liner wall from the primary zone. No trace of the vortex generated by the inter-mixing of the dilution jets and the crossflow is found in this section. The velocity of the outflow at the exit plane is higher than those measured in the previous two sections indicating that most outflow is routed via planes that are further away from the fuel injector. Due to the blockage of the laser beams by the fuel injector, a small part of the flowfield (approximately where the injector was located) was not able to be measured.

### 6.3.2 Total Velocity $V_t$

**Cold Flow Section 1 (Fuel Injector Plane)**

The contour and the surface plots of the total velocity $V_t$ are shown in Figs. 79 and 80, respectively. A small vortical flow with velocities less than 4 m/s is seen near the ignitor. The wall layer flow along the liners has a slightly higher velocity since air enters the combustor through the wall tangential louvres. An even smaller region with very small velocities is also found just above the centerline and very close to the front dome where the splashing fuel jet pushes the downward flow to change its flow direction and recirculate back up along the front dome liner right underneath the ignitor. In the primary zone, the dominating feature is the fuel jet incoming at a total velocity of about 107 m/s. The core velocity of this fuel jet is seen decaying quite rapidly as it dashes through the combustor and splashes itself at the front dome bottom liner with total velocity magnitudes of less than 10 m/s. The decay of the fuel jet upstream and downstream are also clearly seen. Upstream, the surrounding air is flowing in the same direction as the fuel jet along the longitudinal axis, thus the velocity gradient in the shear layer of the fuel jet is moderate. However, higher total velocity gradient occurs downstream which is caused by the influence of the opposing primary jet.
In the vortex just downstream of the fuel jet, the total velocity is less than 6 m/s. The measured velocity near the bottom louvre slightly downstream of the fuel jet is 10 m/s. The primary jet enters with a total velocity of \( \sim 56 \text{ m/s} \) and decreases in magnitude rapidly initially near the inlet. Comparing the contour line shape of this primary jet with the one in cold flow without fuel-injection, this jet has an almost vertical flow trajectory and its velocity magnitude decays much slower far downstream. These can be expected because the fuel jet upstream creates a low pressure region near its vicinity which favors entrainment of the surrounding air into the fuel jet. The bulk of the primary flow is transported to the neighboring planes by the fuel jet, hence the primary jet experiences a minimum amount of crossflow as it cuts across the combustor. The upstream part of the primary jet is eventually entrained into the fuel jet. The portion of the fuel jet that made it to the fuel injector has an impact velocity of about 4 m/s, which is almost half of the impact velocity of the fuel jet at the bottom liner. Some portions of the primary jet mix with the air along the top liner and air from the thumb-nail louvre. The portion that reaches the secondary zone has a velocity magnitude of about 6 m/s before its crossflow with the top dilution jet. The rest of the primary jet recirculates in the secondary recirculation zone, which is much larger in size than in the cold flow without injection case. The velocity magnitudes within the secondary recirculation varies from 0.62 m/s at the center to about 8 m/s near the bottom liner.

In the secondary zone, the top dilution jet enters at \( \sim 58 \text{ m/s} \), which is nearly identical to the inlet velocity of the primary jet. This agreement is expected since the density of the airflow and the pressure drop across the combustor are the same for both the primary and dilution holes. Although the size of these holes are different, their discharge coefficients do not differ appreciably. In one inlet diameter downstream, it is seen that the velocity decay rate of the fuel jet is slower than that of the dilution jet, and in turn the velocity decay rate of the dilution jet is slower than that of the primary jet. These correspond very well with the size of the inlet since more mass flow enters with a larger size inlet and consequently higher momentum which decays in a slower rate. Because of the crossflow of the top dilution jet with the oncoming primary flow, higher velocity gradient is found upstream of the jet. Just below the centerline of the combustor, the rapid turning of the flow creates a local minimum velocity of 2.4 m/s. Near the bottom liner in the secondary zone, the flow velocity is about 10 m/s. In the exit plane, the velocity magnitude is about 10 m/s in the upper part of the combustor and decreases to about 5 m/s near the bottom.

Hot Flow Section 1 (Fuel Injector Plane)

The contour and surface plots of the combusting flow in Section 1 are shown in Figs. 81 and 82. The small vortical flow in front of the ignitor has moved upward and closer to the front dome because of the expansion of the combusting fuel jet downstream. It has a slightly higher velocity here than in the cold flow. The wall layer also has a higher flow velocity of about 10 m/s along the top of the front dome liner. Underneath the ignitor are two small vortices with velocities of about 3 m/s. The dominating fuel jet enters at 82 m/s, cuts across the primary zone as it decreases in magnitude, and splashes onto the bottom of the front dome liner. The impact velocity is about 20 m/s, which is about five times of that in the cold flow. Near the fuel jet exit upstream, entrainment is clearly seen delayed to about one fuel nozzle inlet diameter.
downstream because of the local *laminarization* of the fuel jet due to the acceleration of the jet flow and the combustion process increasing the kinematic viscosity of the gas mixture. This *laminarization* prevents the mixing process of the fuel jet with the surrounding air. Above the centerline of the combustor upstream of the fuel jet is a local minimum of 0.84 m/s. This occurs at a location where the two small vortical flows split. At the bottom liner upstream of the fuel jet, the vortex created by the splashing has velocity magnitude ranges from 6 to 18 m/s instead of only about 4 m/s as in the cold flow case. At the bottom liner downstream of the fuel jet, the triangular-shaped vortex has velocities varying from near zero to 10 m/s (compared with 0 to 6 m/s in the cold flow). Above the liner near the bottom louvre, the flow velocity is about 18 m/s, which is almost twice of that found in the cold flow. This increase is due to the expansion of the fuel jet, causing higher impact velocity at the bottom liner and resulting in higher side-flow velocity created by the splashing action. The core of the fuel jet is seen decreasing more gradually than in the cold flow. Similar to the cold flow case, higher velocity gradient occurs downstream of the fuel jet due to the interaction with the primary jet.

The inlet velocity of the primary jet is ~55 m/s and its decay rate is much lower than that in the cold flow. The flow trajectory of the primary jet in penetrating across the combustor is more vertical due to the slower decay rate caused by the low density of the gas mixture inside the combustor. The part of the primary jet flow that reaches the fuel injector dome has a velocity of about 6 m/s. The part of the primary jet that enters the secondary zone has a velocity of about 8 m/s. The rest of the primary jet flow recirculates into the secondary recirculation zone which has velocities very similar to that in the cold flow. In the secondary zone, the top dilution jet enters at ~57 m/s and is seen decaying down to 3 m/s about the centerline of the combustor where the flow turns drastically downstream. The flow velocity near the bottom liner is slightly higher than that in the cold flow. In general, the exit velocities are higher than those of the cold flow by 2 m/s, except about the centerline where an increase of about 4 m/s is found.

*Hot Flow Section 2*

Figures 83 and 84 show the contour and surface plots of the total velocity in Section 2. The primary vortical flow in the upper part of the primary zone has a velocity range of near zero in the center of the vortex to about 8 m/s along the top liner and just above the fuel jet. Since the fuel jet enters this section at about two fuel injector inlet diameter downstream along the jet trajectory, $V_t$ is only 18 m/s near the injector inlet and decreases to about 6 m/s at a distance of one fuel injector inlet diameter, and then increases gradually as the jet starts penetrating the region. The local maximum total velocity in the fuel jet is only 27 m/s compared with 37 m/s in Section 1. However, the impact velocity of the fuel jet at the bottom liner of the front dome is similar to that in the fuel injector plane at ~20 m/s. The velocity gradient in the shear layer on both sides of the fuel jet is about the same. Due to the splashing upstream of the jet, the two small vortices near the front louvre have a velocity range from about 4 to 17 m/s. The vortical flow generated by the splashing of the fuel jet, the bottom louvre flow and the vertically-inclined primary jet have velocities from near zero in the center...
to about 12 m/s. The measured velocity near the bottom louvre is 16 m/s, which is only 2 m/s smaller than that seen in Section 1.

Since this section is very close to the neighboring primary jet inlet hole, a maximum velocity of 44 m/s near this jet inlet is measured. The trajectory of this primary jet is now directed more downstream, which is most probably due to the lesser amount of suction effect from the fuel jet near its inlet. The velocity gradients on both sides of the primary jet have also dropped compared with those in Section 1. The part of the flow from the primary jet that reaches the top liner around the thumb-nail louvre has a velocity of only 3 m/s. Some primary flow enters the secondary zone above the secondary recirculation at about 10 m/s. The secondary recirculation has velocities from 0.5 to 8 m/s, which is not too different from those in Section 1, but the local minimum of this vortex has shifted downward at a minute amount downstream. The bottom dilution jet enters the combustor at $\sim$58 m/s, which is nearly the same as the velocity of the top dilution jet in Section 1. Higher velocity gradient is seen upstream because of the strong cross flow between the two opposing dilution jets. A local minimum velocity region is seen above the centerline upstream of the bottom dilution jet where the primary flow splits and the top and bottom dilution jets' velocities cancel out almost entirely. At the exit plane, higher velocities are seen in the upper half of the combustor since most of the flows leave near the top liner. The velocities at the exit plane vary from about 5 to 20 m/s.

**Hot Flow Section 3**

The contour and surface plots of $V_t$ in Section 3 are depicted in Figs. 85 and 86. Since the fuel jet has ceased to appear, the primary zone is much simpler in this section. The primary vortex near the front dome above the centerline has a velocity varying from zero in the center to about 10 m/s along the top and front dome liners. Downstream of the primary vortex, the bulk of the primary flow goes towards the secondary zone with velocities between 8 to 14 m/s. Because there is a wake flow on either side of the fuel injector also in the tangential direction, a low velocity zone is found near the top liner where the fuel injector would be located. The local maximum of the primary jet is located slightly above the bottom liner with a value of 25.9 m/s. The velocity magnitude then decreases downstream to near zero in the elliptic secondary recirculation zone. The velocity gradient upstream of the primary jet is almost non-existent and that downstream is much more moderate compared with that found in other sections. At the bottom liner near the bottom louvre, the velocity gradient is smaller than those found in other sections with the fuel jet and the highest velocity measured is 16 m/s near the liner, which is similar to that in Section 2. Under the top double-band and the thumb-nail louvres, the velocity is below 6 m/s.

The primary flow enters the secondary zone almost across the entire cross-section of the combustor with velocity varying from about 4 to 14 m/s, the maximum velocity being generated by the primary jet at about the centerline of the combustor. The bottom dilution jet enters at $\sim$55 m/s and penetrates across the flowfield. Because the oncoming primary flow has a higher velocity and consequently a stronger crossflow, this bottom dilution jet does not display a deep penetration as the bottom dilution jet in Section 2. The flow leaving the
secondary zone along the upper half of the sector has velocities between 18 to 26 m/s. In the wake flow of the bottom dilution jet, the flow velocity varies from 8 to 16 m/s. In the upper half of the exit plane, the total velocity magnitude is nearly constant at 20 m/s.

6.3.3 Velocity  \( \bar{u} \)

**Cold Flow Section 1 (Fuel Injector Plane)**

The contour and surface plot of the longitudinal velocity \( \bar{u} \) are shown in Figs. 87 and 88. Most of the flow in the primary zone has a negative \( \bar{u} \) velocity. The \( \bar{u} \) velocity along the top liner is about \(-5\) m/s and it decreases toward the center of the vortex. As the top flow reaches the front dome, the \( \bar{u} \) velocity decreases. In the front dome, a small region of positive \( \bar{u} \) is seen just below the ignitor with magnitudes less than 2 m/s. The fuel jet is entering at \(-72\) m/s and is seen decaying as it penetrates across the primary flow toward the bottom liner. The gradient of \( \bar{u} \) is higher downstream of the fuel jet. The splashing of the fuel jet downstream and the flow from the bottom louver cause \( \bar{u} \) to become positive again. The primary jet enters the combustor with \( \bar{u} \) at \( \sim 10\) m/s and it flows mainly downstream since \( \bar{u} \) is almost entirely positive here, except for the small part of the flow that is entrained into the fuel jet. A local maximum of 8 m/s in the primary jet flow is seen between the fuel injector and the secondary recirculation zone. From the center of the secondary recirculation, the magnitude of \( \bar{u} \) increases from zero to about \(-10\) m/s near the bottom liner. The flow near the thumb-nail louver is near zero. The \( \bar{u} \) velocity of the primary flow entering the secondary zone near the top liner is about 6 m/s. In the secondary zone, the top jet enters the combustor with \( \bar{u} \) at \( \sim -11.1\) m/s. The gradient of \( \bar{u} \) is higher upstream than downstream of the top dilution jet because of the cross flow with the oncoming primary flow. The interaction of the opposing dilution jets creates some very interesting \( \bar{u} \) contours in the secondary zone. Upstream of the top dilution jet about the centerline, a negative \( \bar{u} \) region with a local minimum of \(-5.9\) m/s exists. At the exit plane, \( \bar{u} \) is fairly constant at about 9 m/s in the upper portion and increases to about 11 m/s about the centerline. After a decrease of \( \bar{u} \) in the lower portion to about 4 m/s, it increases rapidly to 7 m/s near the bottom liner.

**Hot Flow Section 1 (Fuel Injector Plane)**

The contour and surface plots of velocity \( \bar{u} \) in Section 1 of the combusting case are shown in Figs. 89 and 90. In the primary zone, higher longitudinal flow velocity magnitudes of \( \bar{u} \) are found upstream of the fuel jet. The magnitude of \( \bar{u} \) along the top liner is about \(-10\) m/s instead of \(-5\) m/s as in the cold flow. The positive \( \bar{u} \) region in front of the ignitor has a higher local maximum of 5.87 m/s instead of 1.5 m/s in the cold flow. The increase in velocity is due to the decrease in density of the gas mixture because of combustion. However, since the setup condition was to maintain the pressure drop across the combustor to be constant, the mass flow into the combustor has dropped slightly. The velocity \( \bar{u} \) at the fuel injector inlet has decreased to \( \sim -52\) m/s due to the decrease in the total mass flow and the distribution of mass flow according to the local gas mixture temperature. The decay rate of the fuel jet
is more moderate than in the cold flow case. The splashing at the bottom liner of the front dome is nearly 20 m/s instead of about 6 m/s. The velocity gradient both upstream and downstream of the injector inlet are higher than those in the cold flow. At the bottom liner just downstream of the bottom louvre, the flow velocity is now 25 m/s compared with about 10 m/s in the cold flow.

The primary jet enters at ~7 m/s instead of ~10 m/s at the bottom liner across from the fuel injector. A local maximum of 8.8 m/s exists downstream in the shear layer. The primary flow impinging on the fuel injector also has higher velocity than that in the cold flow. Another local maximum of 11.7 m/s is found in the primary flow downstream of the fuel injector dome. The primary flow that enters the secondary zone near the top liner has velocity \( u \) of about 10 m/s. In the secondary recirculation zone behind the primary jet, \( u \) varies from zero near the centerline to about \(-10\) m/s above the bottom liner. In the secondary zone, the top dilution jet enters with \( u \) at \(-9\) m/s compared with \(-12\) m/s in the cold flow. This decrease in \( u \) at the top dilution jet inlet is due to the higher \( u \) momentum of the oncoming crossflow from the primary zone. Also, higher \( u \) gradient is seen upstream of the dilution jet because of the cross flow. The negative \( u \) velocity region in the secondary zone near the centerline of the combustor has a velocity magnitude of 7 m/s, which is about 1 m/s larger than that in the cold flow. At the bottom liner, the highest \( u \) is about 16 m/s instead of about 12 m/s found in the cold flow. The velocity \( u \) is much higher at the exit plane with a magnitude of about 13 m/s, compared to 9 m/s in the cold flow. The rest of the exit velocities are also higher. Overall, the exit velocity of the combusting flow is about 3 to 4 m/s higher than those in the cold flow.

**Hot Flow Section 2**

The contour and surface plots of velocity \( u \) in Section 2 are shown in Figs. 91 and 92. In the primary zone, the positive \( u \) velocity region is more extended. Along the top liner, the velocity \( u \) is slightly smaller than that in Section 1 at about 8 m/s. The positive \( u \) region now engulfs a large region upstream of the fuel jet with a local maximum of 9.4 m/s in the primary vortex. A small negative region exists just above the centerline where an oblong recirculation is found along the front dome liner with near zero velocity. At the fuel injector inlet, the measured \( u \) is only ~ \(-9\) m/s instead of ~ \(-52\) m/s as in Section 1, since this section is located at one fuel injector inlet diameter away from the center of the fuel injector. The fuel jet appears as a negative region with a local minimum of \(-23.2\) m/s along the axis of the fuel jet trajectory. The velocity gradients on both sides of the fuel jet are quite similar to those in Section 1. The fuel jet splashes on the bottom liner at \sim\sim \(-19\) m/s, which is basically no different from that in Section 1. The flow near the bottom louvre has decreased in strength with \( u \) about 15 m/s compared with 25 m/s in Section 1.

The primary jet near the bottom liner is drawn by the fuel jet vortex upstream and is pushed by the recirculating flow in the secondary recirculation to move in the negative longitudinal direction. As mentioned in the discussion of \( V_t \), more mass is leaving the primary zone in Section 2 than in Section 1. The local maximum of \( u \) in the primary jet is 14.5 m/s compared with 11.7 m/s in Section 1. In the secondary recirculation zone, the zero contour
level lies lower and closer to the bottom liner than those seen in Section 1, and the highest magnitude of $\bar{u}$ in the secondary recirculation is along the bottom liner at about 9 m/s, which is the same as that in Section 1. Hence a higher velocity gradient exists in the secondary recirculation zone. In the secondary zone, the bottom dilution jet enters with $\bar{u}$ at $\sim$9 m/s and increases slightly to about 15 m/s near the top liner. Upstream of the dilution jet is a negative velocity region with a local minimum of $-5.2$ m/s, which is created by the separation of the primary jet flow, as well as the interaction of the two opposing dilution jets. At the exit, the outflow $\bar{u}$ is about 13 m/s near the top liner. Then it increases to about 17 m/s about the center, and decreases to about 4 m/s in the wake of the dilution jet. A layer of higher velocity $\bar{u}$ of 7 m/s exists along the bottom liner. In general, because of the larger amount of mass leaving the combustor via this section, $\bar{u}$ is higher here than in Section 1 in the upper part of the exit plane.

**Hot Flow Section 3**

Figures 93 and 94 depict the contour and surface plots of the velocity $\bar{u}$ in Section 3 of the combusting case. In contrast to the strongly negative $\bar{u}$ velocity found in the primary zone of Section 1 (fuel injector plane), $\bar{u}$ is predominantly positive here in Section 3. The primary vortex is located very close to the front dome and the top liner. The center of this vortex lies around the center of the zero contour line. The velocity $\bar{u}$ increases from this center to about 13 m/s near the top liner. Downstream of this vortex, $\bar{u}$ increases gradually to about 12 m/s before merging with the primary jet. At the bottom liner, the velocity $\bar{u}$ near the bottom louvre reaches about 16 m/s, which is the same as that seen in Section 2. At the location of the primary, $\bar{u}$ decreases to below 10 m/s because the jet enters vertically. As the primary jet bends downstream by the oncoming primary flow, $\bar{u}$ increases to a maximum of 20.1 m/s above the secondary recirculation near the centerline.

In the secondary recirculation zone, $\bar{u}$ varies from about 6 m/s to $-6$ m/s above the bottom double band louvres. Also, a higher velocity gradient is found downstream of the primary jet. The majority of primary flow enters the secondary zone at the upper half of the combustor with $\bar{u}$ at about 11 m/s, which is similar to that in Section 2. The bottom dilution jet enters with $\bar{u}$ at $\sim-4.1$ m/s and then faces the crossflow which causes $\bar{u}$ to increase further downstream. As the dilution jet flows towards the top liner, it forces the primary flow to speed up in the upper part of the combustor, resulting in a local maximum $\bar{u}$ of 23.5 m/s near the top liner and about 21.6 m/s at the centerline. In the wake of the bottom dilution jet, $\bar{u}$ decreases to near zero just downstream of the jet. At the exit plane, $\bar{u}$ increases from 16 m/s near the top to a peak of 19 m/s at the centerline and then decreases to about 9 m/s in the wake flow. At the bottom liner, $\bar{u}$ increases up to 12 m/s, which is higher than that in the other two sections. Overall, velocity $\bar{u}$ at the exit plane is higher in this section than the other sections, thus indicating more mass outflow in sections that are further away from the fuel injector plane.
6.3.4 Velocity $\bar{v}$

**Cold Flow Section 1 (Fuel Injector Plane)**

The contour and surface plots of the velocity $\bar{v}$ in the cold flow of Section 1 are shown in Figs. 95 and 96. In the primary vortex close to the ignitor, because the flow along the top liner is nearly horizontal, $\bar{v}$ is near zero. As the flow turns towards the front dome at the ignitor, $\bar{v}$ becomes negative and increases to $-8$ m/s in front of the ignitor. However, directly underneath the ignitor is an oblong recirculation with $\bar{v} \sim 1$ m/s. The fuel jet is seen dominating the primary zone. The fuel jet has an inlet velocity of $-79$ m/s in $\bar{v}$. It decreases slightly and then increases up to $45$ m/s and then decreases quite moderately to $7$ m/s as it impinges on the bottom liner. A higher velocity gradient exists downstream of the fuel jet than upstream due to the oblique crossflow of the primary jet with the fuel jet. As the fuel jet splashes on the bottom liner, two vortices are formed. The one upstream is located just below the front dome louvre with a high of $5$ m/s, and it is smaller in size than the one downstream. The second vortex is downstream of the fuel jet with $\bar{v}$ changing sign from negative to positive in the fuel jet, and it reaches $5$ m/s before encountering the primary jet.

At the inlet of the primary jet, $\bar{v}$ is $\sim 55$ m/s and it flows almost vertically towards the fuel injector as it decays gradually to $4$ m/s at the dome of the fuel injector. Since there is flow from the thumb-nail louvre behind the flow injector, $\bar{v}$ becomes negative. The part of the primary flow that enters the secondary zone is near horizontal, thus $\bar{v}$ is low at $2$ m/s near the top liner. Behind the primary jet is a large region of negative $\bar{v}$ because of the big secondary recirculation in this zone. $\bar{v}$ is relatively small in this vortex. In the secondary zone, the top dilution jet enters the combustor with $\bar{v}$ at $-49$ m/s. It then decays rapidly to $-10$ m/s at the centerline of the combustor. The rolling of the dilution jet flow downstream causes $\bar{v}$ to drop further in magnitude and reaches zero at the turning point. The dilution jet flow loses most of its momentum because more mass has been injected into the combustor through the fuel injector, hence higher momentum is transported by the primary flow into the secondary zone and the dilution jets experience stronger crossflow which results in reduced penetration of the dilution jet. The effect of this crossflow and the interaction of the two opposing dilution jets are responsible for the lower velocity region in the lower part of the combustor in the secondary zone. Similar to the $\bar{u}$ velocity, a higher velocity gradient in $\bar{v}$ is found upstream of the top dilution jet. At the exit plane, $\bar{v}$ is fairly constant at below $-1$ m/s in the upper part, reaches $-3$ m/s below the centerline and then decreases slightly in the wake flow near the bottom liner.

**Hot Flow Section 1 (Fuel Injector Plane)**

Shown in Figs. 97 and 98 are the contour and surface plots of the velocity $\bar{v}$ in Section 1 of the hot flow. Because of the expansion of the flow due to combustion, the vortices are enhanced and the flow velocities are also increased in magnitude. In the primary zone, the positive $\bar{v}$ regions are appreciably enlarged compared with those in the cold flow. The primary vortex has moved slightly away from the front dome and the top liner. The local maximum
of $\bar{v}$ in the primary vortex has increased to 3.5 m/s. At the face of the ignitor, $\bar{v}$ is $-10$ m/s compared with $-8$ m/s in the cold flow. Although the dominating fuel jet enters at a lower velocity $\bar{v}$ of $\sim -63$ m/s than in the cold flow, the velocity decays at a much slower rate along the axis of the fuel jet due to the expansion effect of the fuel jet with temperature. The fuel jet impinges on the bottom liner with $\bar{v}$ at $-17$ m/s instead of $-7$ m/s in the cold flow. The upstream vortex created by the splashing is also seen with larger positive $\bar{v}$ region. The local maximum in this vortical flow is about 15 m/s instead of 5 m/s. In the vortex downstream of the fuel jet splashing, $\bar{v}$ has a local peak of 10 m/s instead of 7 m/s.

The primary jet enters with $\bar{v}$ at $\sim 53$ m/s, which is nearly the same as in the cold flow. However, similar to the fuel jet, the decay rate of the primary jet along its trajectory axis is slower than that in the cold flow. The velocity $\bar{v}$ of the primary jet reaching the fuel injector is 6 m/s. Since most of the primary flow leaving the zone is at the upper half of the combustor and almost horizontal, $\bar{v}$ is zero underneath the double-band louvres at the top liner. The $\bar{v}$ velocities in the secondary recirculation are quite similar to those in the cold flow except that the rotation of the velocity vectors is tighter. In the secondary zone, the top dilution jet enters with $\bar{v}$ at $\sim -50$ m/s, which is almost identical to that of the cold flow. The jet is seen decaying as it cuts across the flowfield. However, the depth of penetration here in the hot flow is slightly less than that in the cold flow. This is because of the generally higher flow velocity of the oncoming flow that engages in the crossflow with the dilution jet. The positive region at the turning point of the jet flow has higher velocity $\bar{v}$ of about 3 m/s. At the exit plane, $\bar{v}$ is slightly higher in the upper part and about the same in the rest of the cross-section compared with those in the cold flow.

**Hot Flow Section 2**

The contour and surface plots of the velocity $\bar{v}$ in Section 2 are shown in Figs. 99 and 100. In the primary zone, the primary vortex has lesser restriction from the fuel jet in preventing it from extending further downstream. The center of the primary vortex lies on the zero contour level in front of the ignitor. In comparison with Section 1, the negative velocity $\bar{v}$ region is larger. The highest $\bar{v}$ measured on the face of the ignitor is still $-10$ m/s as in Section 1. The local maximum of $\bar{v}$ in the primary vortex is 3.2 m/s, which is almost no different from the one in the fuel injector plane. Upstream of the fuel injector is a large region of positive $\bar{v}$ with magnitudes between 1 and 3 m/s. This is a low velocity region because the primary vortex terminates in this area and begins to reverse its direction back into the front end of the combustor. The emergence of the fuel jet into this section is seen clearly in the primary zone with a maximum velocity $\bar{v}$ of 20 m/s near the bottom liner. The fuel jet impinges on the bottom liner at $-13$ m/s, which is similar to the magnitude found in Section 1. Also, the vortex upstream of the fuel jet created by the splashing effect is nearly the same size and magnitude as that in the previous section. However, the vortex downstream of the fuel jet has lower velocity gradient due to the fact that the primary jet is not as strong as that in the fuel injector plane.

The primary jet trajectory is oriented more downstream. A local maximum of $\bar{v}$ in the primary jet is found just above the bottom liner near the jet inlet with a value of 43.5 m/s.
The primary flow that enters the secondary zone has \( \bar{v} \) of about 5 m/s, since the flow is less horizontal here. The secondary recirculation is very similar to that in Section 1 except that it has moved closer to the bottom liner because more mass flows towards the secondary zone in this section and thus forces the secondary recirculation further downward. In this recirculation, a local minimum of \(-13.2 \text{ m/s}\) is found upstream of the bottom dilution jet. At the inlet of the bottom dilution jet, \( \bar{v} \) is \( \sim 53 \text{ m/s} \). Much higher velocity gradient exists on the upstream because the secondary recirculation is flowing in exactly the opposite direction as the bottom dilution jet due to the influence from the top dilution jet. As the dilution jet cuts across the flowfield, velocity \( \bar{v} \) decreases. Eventually along the top liner, \( \bar{v} \) is about 1 m/s. At the exit, \( \bar{v} \) increases from 1 m/s to 10 m/s in the upper part and then decreases to 2 m/s in the lower half of the combustor.

**Hot Flow Section 3**

Figures 101 and 102 show the contour and surface plots of the velocity \( \bar{v} \) in Section 3. In the primary zone, except the front end of the combustor, the velocity \( \bar{v} \) in the entire primary flow is positive. The \( \bar{v} \) velocity increases from \(-11 \text{ m/s}\) along the front dome to about 9 m/s before encountering the primary jet downstream. Since the center of the primary vortex must lie on the zero contour line, one can realize how close to the front dome is the primary vortex in the combustor under the influence of the fuel jet. Within the primary jet, a local maximum in \( \bar{v} \) at 17.9 m/s is seen. On either side of the fuel injector in the tangential direction, there is a wake region. Approximately where the fuel injector is situated in this section, the wake flow around the injector and the flow from the thumb-nail louvre has negative \( \bar{v} \) velocity near the top liner. The highest velocity measured there is 6 m/s. The secondary recirculation region has decreased significantly in this section compared to those in the other two sections. A minimum of \(-6 \text{ m/s}\) is found just before the bottom dilution jet which enters at \( \sim 55 \text{ m/s} \). The velocity gradient upstream of this dilution jet is much higher than downstream. The decay of the dilution jet can be clearly observed with the depth of penetration significantly less than that present in the cold flow. Near the top liner, the leaving primary flow is pressed toward the top and flows toward the exit plane with a velocity \( \bar{v} \) less than 10 m/s. At the exit plane, \( \bar{v} \) decreases from about 5 m/s at the top liner to near zero below the centerline.

### 6.4 Reynolds Stresses \( \overline{u'^2}, \overline{v'^2} \) and \( \overline{u'v'} \)

#### 6.4.1 Normal Stress \( \overline{u'^2} \)

*Cold Flow Section 1 (Fuel Injector Plane)*

The contour and surface plot of the normal stress \( \overline{u'^2} \) in the longitudinal direction for the cold flow in Section 1 are shown in Figs. 103 and 104. In the primary zone, \( \overline{u'^2} \) is the lowest in front of the ignitor at 5 m²/s² because the cooling air layer is not disturbed by other flow along the liner. Within the primary vortex, \( \overline{u'^2} \) increases in magnitude from the liner
toward the center. The fuel jet enters with $u^2$ at $\sim 730 \text{ m}^2/\text{s}^2$. It then decays as it cuts across the primary zone to about 100 $\text{m}^2/\text{s}^2$ at the bottom liner. The gradient of the normal stress is higher near the fuel jet inlet than further downstream of the jet trajectory. The gradient upstream is slightly higher than that downstream. In the small vortex downstream of the fuel jet, $u^2$ has dropped to about 10 $\text{m}^2/\text{s}^2$.

At the inlet of the primary jet, $u^2$ is $\sim 44 \text{ m}^2/\text{s}^2$. In the core of the primary jet, a local maximum of 118 $\text{m}^2/\text{s}^2$ is observed just slightly downstream of the primary jet inlet. The remaining part of the primary jet flow has very low normal stress $u^2$ predominantly below 30 $\text{m}^2/\text{s}^2$. In the region of the primary flow that enters the secondary zone near the top liner $u^2$ is about 15 $\text{m}^2/\text{s}^2$. In the secondary recirculation, $u^2$ is below 25 $\text{m}^2/\text{s}^2$. In the secondary zone, the top dilution jet enters with $u^2$ at $\sim 110 \text{ m}^2/\text{s}^2$. The normal stress then increases gradually as the dilution jet penetrates into the combustor, and reaches a maximum value of 162.5 $\text{m}^2/\text{s}^2$ about the centerline of the combustor. A higher gradient is found upstream due to the crossflow of the dilution jet with the primary flow. From this contour plot, the turning of the dilution jet in the secondary zone is not obvious. Rather, the shape of the contours behaves similar to that in the cold flow. At the exit plane, $u^2$ increases from 30 $\text{m}^2/\text{s}^2$ near the top liner to a high of 50 $\text{m}^2/\text{s}^2$ and then decreases back to about 25 $\text{m}^2/\text{s}^2$ near the bottom liner.

**Hot Flow Section 1 (Fuel Injector Plane)**

Figures 105 and 106 depict the contour and surface plots of the normal stress $u^2$ in Section 1 of the hot flow. In the primary zone, $u^2$ has increased almost everywhere due to the enhanced turbulence motion with combustion. The primary vortex has a general increase of about 35% in $u^2$ with combustion. The fuel jet has an inlet $u^2$ of $\sim 680 \text{ m}^2/\text{s}^2$. In contrast to the results obtained in the cold flow, $u^2$ does not decay monotonously, but decreases initially near the jet inlet and then increases to a peak of 348.2 $\text{m}^2/\text{s}^2$ before decreasing again. The normal stress of the fuel jet near the injector inlet is smaller than that in the cold flow; however, it is higher by about 100% from the location of the peak value down to the bottom liner. The shear layer of the fuel jet can be seen extended further outward than in the cold flow. The normal stress is also slightly higher in the vortex downstream of the fuel jet.

The primary jet enters with $u^2$ of $\sim 52 \text{ m}^2/\text{s}^2$ and reaches a local maximum of 132.7 $\text{m}^2/\text{s}^2$ slightly downstream of the jet's inlet. The entire primary flow has $u^2$ slightly higher in this combusting flow than in the cold flow. However, the secondary recirculation zone shows very little difference from the cold flow case, except that this zone is pressed closer to the bottom because of the expanding hot flow from upstream. The primary flow that leaves the primary zone now has $u^2$ at about 25 $\text{m}^2/\text{s}^2$ instead of 15 $\text{m}^2/\text{s}^2$ as in the cold flow. In the secondary zone, the top dilution jet enters the combustor at 140 $\text{m}^2/\text{s}^2$. As it cuts across the flowfield, it reaches a local maximum of 198.4 m/s at about the centerline of the combustor which is 22% higher than that in the cold flow. At the exit, $u^2$ is also higher with the combusting case than with the cold flow. The normal intensity $u^2$ increases from 30 $\text{m}^2/\text{s}^2$ near the top liner to a high of almost 70 $\text{m}^2/\text{s}^2$ at the centerline, and then decreases to 40 $\text{m}^2/\text{s}^2$ in the bottom wake flow.
Hot Flow Section 2

The contour and surface plots of $\bar{u}^2$ in Section 2 are shown in Figs. 107 and 108. The normal stress $\bar{u}^2$ in the primary vortex has decreased in magnitude compared with Section 1 because the influence of the strong fuel jet does not enter into play until further downstream. The vortex thus takes up the space near the injector inlet before the fuel jet emerges in this section. The intensity $\bar{u}^2$ in the primary vortex varies from 6 m$^2$/s$^2$ near the front top liner to 25 m$^2$/s$^2$ upstream of the fuel injector. In front of the fuel injector, there is a combination of flows (see Fig. 77). First, there is a small amount of flow exiting from the fuel injector which is very likely having a component in the tangential direction. Second, the primary jet turns sharply before reaching the fuel injector and is entrained into the fuel jet. Third, some of this primary flow crosses the fuel jet and merges with the primary vortex upstream. Fourth, the primary vortical flow crosses head-on with the oncoming flow from downstream, hence forcing it to turn and recirculate back into the front dome. Therefore an increase of $\bar{u}^2$ near the fuel injector inlet is observed. As the fuel jet enters this plane, $\bar{u}^2$ increases rapidly and reaches a maximum value of 309.1 m$^2$/s$^2$ which is comparable to that in Section 1 and the location is moved slightly downstream possibly because the strength of the portion of the fuel jet in this section is not as strong as the core flow found in Section 1; the primary jet is also weaker in this plane. In the vortex downstream of the fuel jet, $\bar{u}^2$ decreases in magnitude as the vortical flow mixes with the primary jet and flows upward towards the thumb-nail louvre. A local minimum of 8.4 m$^2$/s$^2$ is found about the centerline in this vortex.

The primary jet enters this plane at $\sim$120 m$^2$/s$^2$ and decays rapidly to 25 m$^2$/s$^2$ before entering the secondary zone. In the secondary recirculation, $\bar{u}^2$ has also dropped slightly in magnitude. In the secondary zone, $\bar{u}^2$ is $\sim$46 m$^2$/s$^2$ at the inlet of the bottom dilution jet and it increases to a local maximum of 150 m$^2$/s$^2$ just above the centerline of the combustor. Further downstream is another maximum of $\bar{u}^2$ at 154.5 m$^2$/s$^2$. These are the double peaks that are characteristics of a free jet as observed by Sislian, Jiang and Cusworth (1988). At the exit, $\bar{u}^2$ increases from about 40 m$^2$/s$^2$ to a high of 94 m$^2$/s$^2$ at the centerline and then decreases to 30 m$^2$/s$^2$ near the bottom liner.

Hot Flow Section 3

Figs. 109 and 110 depict the contour and surface plots of $\bar{u}^2$ in Section 3. Along the front dome liner, $\bar{u}^2$ is the lowest at about 1 m$^2$/s$^2$. Then it increases downstream and reaches a local maximum of 16.3 m$^2$/s$^2$ just below the centerline in the primary zone. Another local maximum of 18.1 m$^2$/s$^2$ is located above the primary jet in the upper part of the flowfield where the primary vortical flow splits with some recirculating back and the rest going downstream. A high gradient of $\bar{u}^2$ is seen where the primary jet appears. At the centerline upstream of the primary jet is a local minimum of $\bar{u}^2$ at 8.2 m$^2$/s$^2$. Downstream of the primary jet, a local maximum of 102.3 m$^2$/s$^2$ is found where the slow moving core of the secondary recirculating flow encounters the high velocity primary jet flow. In comparison with $\bar{u}^2$ in the secondary recirculation of the other two sections, $\bar{u}^2$ is very different in this secondary recirculation. Because this vortex is flattened by the high velocity flow over it, higher magnitude and gradient of $\bar{u}^2$ are found near the bottom liner in the secondary recirculation zone.
The values of $u_{12}$ in the primary flow that crossflows with the bottom dilution jet in the upper half of the combustor is about 20 to 25 m$^2$/s$^2$, which is of similar magnitude as that in the other two sections. In the secondary zone, the bottom dilution jet enters the combustor with $u_{12}$ at 56.7 m$^2$/s$^2$ and it increases to two peaks. One local maximum is 145 m$^2$/s$^2$ and the other located slightly downstream is 240 m$^2$/s$^2$. These double peaks are observed also in the previous section. The double peaks here are uneven probably because of the distortion by the crossflow of the bottom dilution jet with the primary flow upstream. In the exit plane, $u_{12}$ increases from about 40 m$^2$/s$^2$ near the top liner to 100 m$^2$/s$^2$ above the centerline and then decreases to 30 m$^2$/s$^2$ above the bottom liner.

### 6.4.2 Normal Stress $v_{12}$

#### Cold Flow Section 1 (Fuel Injector Plane)

Depicted in Figs. 111 and 112 are the contour and surface plots of the normal stress $v_{12}$ in Section 1 of the cold flow. In the primary vortex, $v_{12}$ is low, about 5 to 20 m$^2$/s$^2$. At the inlet of the fuel jet, $v_{12}$ is extremely high at ~1250 m$^2$/s$^2$, and it decays with high gradient initially as it penetrates into the primary zone. When the fuel jet impinges onto the bottom liner, $v_{12}$ decreases to about 60 m$^2$/s$^2$. The gradient of $v_{12}$ downstream is seen to be higher than that upstream due to the interaction with the primary jet flow. In the vortex downstream of the fuel jet, $v_{12}$ reaches similar low levels as that in the primary vortex. At the inlet of the primary jet, $v_{12}$ is ~190 m$^2$/s$^2$ and is decaying very fast initially before attaining a stable level of about 20 m$^2$/s$^2$ in most parts of the primary jet flow. Within the secondary recirculation, $v_{12}$ is also as low as those in the other vortices of about 10 m$^2$/s$^2$. The value of $v_{12}$ in the primary flow entering the secondary zone is 15 m$^2$/s$^2$. In the secondary zone, the top dilution jet enters the combustor with $v_{12}$ at ~520 m$^2$/s$^2$. Higher gradient of $v_{12}$ is found upstream since the dilution jet crossflows with the primary flow there. About the centerline in the secondary zone, $v_{12}$ reaches a maximum of 686.6 m$^2$/s$^2$ where the top dilution jet is strongly influenced by the crossflow with the primary flow and the interaction with the bottom dilution jet which causes it to turn drastically downstream. At the exit, $v_{12}$ increases from about 20 to 70 m$^2$/s$^2$ rapidly near the top liner and then remains fairly constant at 70 m$^2$/s$^2$ until the centerline of the combustor; it then drops off to about 30 m$^2$/s$^2$ near the bottom liner.

#### Hot Flow Section 1 (Fuel Injector Plane)

Figures 113 and 114 show the contour and surface plots of $v_{12}$ in Section 1 of the hot flow. In the primary vortex, $v_{12}$ is almost twice as high as those in the cold flow with magnitudes ranging from 5 m$^2$/s$^2$ near the ignitor to 30 m$^2$/s$^2$ just upstream of the fuel jet. At the fuel jet inlet, $v_{12}$ is measured to be as high as ~2000 m$^2$/s$^2$. It then decays rapidly as it cuts across the primary zone. Its magnitude decreases to about 130 m$^2$/s$^2$ as it impinges onto the bottom liner. In the upstream vortex of this fuel jet, the magnitudes of $v_{12}$ are much higher than those in the cold flow case and the other vortices found in this plane. In the triangular vortex downstream of the fuel jet, the magnitude of $v_{12}$ does not differ appreciably from the cold flow.
case with magnitudes below 30 m²/s². The primary jet enters at \( \sim 50 \) m²/s² and increases to a local maximum of 156.3 m²/s² before decreasing down to a fairly constant level of about 30 m²/s² in most of its flow downstream. In the secondary recirculation zone, \( \overline{\nu'^2} \) is only slightly higher than that of the cold flow and has values less than 30 m²/s². The primary jet enters the secondary zone with \( \overline{\nu'^2} \) at around 30 m²/s², which is about twice as high as that in the cold flow. At the inlet of the top dilution jet, \( \overline{\nu'^2} \) is higher than in the cold flow at 370 m²/s². It then increases to a local maximum of 845 m²/s² near the centerline where the dilution jet turns drastically downstream toward the exit. This local maximum is about 23% higher than the one in the cold flow at almost the same location. At the exit plane, \( \overline{\nu'^2} \) increases from 50 m²/s² at the top liner to 120 m²/s² just above the centerline and then gradually decreases to about 40 m²/s² near the bottom liner.

**Hot Flow Section 2**

The contour and surface plots of \( \overline{\nu'^2} \) in Section 2 are shown in Figs. 115 and 116. The magnitude of \( \overline{\nu'^2} \) in the primary vortex ranges from about 5 m²/s² near the ignitor to about 20 m²/s² upstream of the fuel jet. These values of \( \overline{\nu'^2} \) in the primary vortex are lower in this plane than in the previous one; the fuel jet is missing near the injector inlet here. However, in the space where the fuel jet is missing, \( \overline{\nu'^2} \) still increases to about 80 m²/s². As the fuel jet emerges, two local maximum of \( \overline{\nu'^2} \) are observed both having similar values of about 220 m²/s². The fuel jet has similar \( \overline{\nu'^2} \) at the point where it impinges on the bottom liner. The gradients on both sides of the fuel jet have decreased in this section compared with Section 1. The normal stress \( \overline{\nu'^2} \) in the vortex upstream of the fuel jet has slightly lower values, whereas \( \overline{\nu'^2} \) changes very little in the vortex downstream of the fuel jet. However, the vortex has expanded slightly. The magnitude \( \overline{\nu'^2} \) in the primary jet near its inlet is \( \sim 190 \) m²/s² and a maximum of about 200 m²/s² is located in the core of this primary jet just a few diameter downstream. Further downstream of this jet flow, \( \overline{\nu'^2} \) has decreased to about 30 m²/s².

In the secondary recirculation, values of \( \overline{\nu'^2} \) are similar but slightly smaller than those in Section 1. Before entering the secondary zone and engaging in the crossflow with the bottom dilution jet, \( \overline{\nu'^2} \) in the primary flow is about 25 m²/s². In the secondary zone, the bottom dilution jet is seen penetrating across the combustor with an inlet \( \overline{\nu'^2} \) value of \( \sim 60 \) m²/s². Then a local maximum of 865 m²/s² is found near the centerline along the axis of the bottom dilution jet. The location of this maximum \( \overline{\nu'^2} \) is slightly lower and the value is about the same as the one in Section 1. Then \( \overline{\nu'^2} \) decreases from this maximum to about 40 m²/s² as it flows toward the top liner. At the exit plane, \( \overline{\nu'^2} \) is high near 140 m²/s² in the upper part of the flow and decreases to about 100 m²/s² in the centerline and about 40 m²/s² near the bottom liner.
Hot Flow Section 3

Figures 117 and 118 show the contour and surface plots of the normal stress $v'^2$ in Section 3. In the primary zone, the contour of $v'^2$ are very much simpler than those in the other two planes where the fuel jet has a significant influence on the flow structure. The values of $v'^2$ are the lowest in this section (below 5 m$^2$/s$^2$) along the front and bottom liners. About the center of the primary vortex, $v'^2$ reaches a local maximum of 22.3 m$^2$/s$^2$. Further downstream of this local maximum, $v'^2$ remains quite low at values between 10 and 14 m$^2$/s$^2$ in the primary zone. However, in the region where the fuel injector is located, $v'^2$ has increased to about 25 m$^2$/s$^2$. This is due to the side-flow induced in the tangential direction on either sides of the fuel injector. The primary jet has a local maximum of 135.3 m$^2$/s$^2$ near the bottom liner. Higher gradient is observed upstream than downstream of this primary jet. The values of $v'^2$ drop to 30 m$^2$/s$^2$ as the primary jet flows toward the top liner and enters the secondary zone. In the secondary recirculation, $v'^2$ varies from about 20 to 2 m$^2$/s$^2$ near the bottom liner. In the secondary zone, the values of m$^2$/s$^2$ in the bottom dilution jet which enters the combustor is 33 m$^2$/s$^2$. In this dilution jet flow, two local maximum are found. The one below the centerline has a value of 568.5 m$^2$/s$^2$, and the one above the centerline is located further downstream of the jet flow with the maximum value of 631.8 m$^2$/s$^2$. Upstream of the dilution jet, the gradient of $v'^2$ is higher due to the crossflow with the primary jet flow. As the primary flow enters and leaves the secondary zone along the top liner, $v'^2$ increases slightly from 40 to 100 m$^2$/s$^2$ as it mixes with the bottom dilution jet. At the exit plane, $v'^2$ increases from 80 m$^2$/s$^2$ at the top to a high of about 110 m$^2$/s$^2$ in the upper part of the flow and then drops down to 20 m$^2$/s$^2$ at the bottom liner.

6.4.3 Shear Stress $u'v'$

Cold Flow Section 1 (Fuel Injector Plane)

The shear stress $u'v'$ in the cold flow of Section 1 is shown in the contour and surface plots of Figs. 119 and 120. In the primary zone, the shear stresses in the primary vortex and along the front top liners are fairly low. A local minimum of about $-2$ m$^2$/s$^2$ is seen in this region. At the fuel injector inlet, $u'v'$ is $\sim$230 m$^2$/s$^2$ and initially it decays rapidly to about 40 m$^2$/s$^2$ in three inlet diameter downstream of the fuel jet axis. Then it increases slightly and reaches local maximum of 56.9 m$^2$/s$^2$ just above the centerline. As this fuel jet strikes the bottom liner, $u'v'$ has decayed down to below 20 m$^2$/s$^2$. Upstream of the fuel jet inlet, $u'v'$ has a local minimum of $-15$ m$^2$/s$^2$ due to the great difference between the velocity of the upper primary flow and the strong fuel jet flow. The gradient of $u'v'$ is generally higher downstream of the fuel jet except near the jet inlet. The small vortex due to the splashing upstream has higher shear stress than the primary vortex, and it has a local maximum of 42 m$^2$/s$^2$. A very high gradient is also observed near the front louvre close to the liner. However, in the larger vortex downstream of the fuel jet, the magnitude of the shear stress is fairly small, at about 2 m$^2$/s$^2$. The primary jet enters from the bottom liner at $u'v'$ of $\sim$40 m$^2$/s$^2$ and $u'v'$ decreases in magnitude as the primary jet flow goes downstream. The sign of $u'v'$ is negative on the
left and positive on the right side of the primary jet axis. Such sign change is typical in a free jet due mainly to the fact that the $\vec{v}$ velocity gradient increases and then decreases when traversing across a vertical jet. Slightly downstream of this primary jet flow, a local maximum of 73.3 $m^2/s^2$ in $u'v'$ is found.

In the secondary recirculation zone, $u'v'$ decreases from the outer edge of about 8 $m^2/s^2$ to slightly below zero near the bottom liner. The primary flow that enters the secondary zone in the upper part of the flowfield has $u'v'$ of 2 $m^2/s^2$. Due to the cross flow of the oncoming primary flow with the strong top dilution jet, a very high gradient of $u'v'$ is present upstream of the dilution jet. The dilution jet has an inlet $u'v'$ of 270 $m^2/s^2$ and the shear stress decays to a local minimum of -6.4 $m^2/s^2$ at the centerline as the dilution jet penetrates across the flowfield and turns downstream sharply near the center of the combustor. In the wake flow below the centerline of the combustor, the gradient of $u'v'$ is fairly high. This may indicate that the shearing actions are caused by flows from the neighboring planes, which are mainly due to the bottom dilution jets. At the exit plane, $u'v'$ increases from about 4 to almost 20 $m^2/s^2$, then decreases to -11 $m^2/s^2$ in the wake and increases up to 5 $m^2/s^2$ near the bottom liner.

**Hot Flow Section 1 (Fuel Injector Plane)**

The shear stress $u'v'$ in the fuel injector plane of the hot flow is shown in the contour and surface plots of Figs. 121 and 122. In the primary vortex, $u'v'$ stresses are higher than those in the cold flow (about double in some locations). The fuel jet enters the flowfield with $u'v'$ at approximately 200 $m^2/s^2$, which is lower than the inlet value in the cold flow. These smaller values of $u'v'$ with the hot flow are due to the temperature rise in the flow which causes the laminar viscosity to increase near the fuel injector inlet, and thus locally *laminarizes* the flow. In about two injector inlet diameter downstream of the fuel jet flow, the magnitudes of $u'v'$ become higher than those in the cold flow for the remaining part of the fuel jet flow. The value of $u'v'$ where the fuel jet impinges at the bottom liner is at least 50 $m^2/s^2$. In the vortical flows upstream of the fuel jet due to splashing, $u'v'$ is also much higher than in the cold flow. The local maximum is 100 $m^2/s^2$. Due to the expansion of the flow because of combustion, the spread of the shear layer downstream of the fuel jet near the bottom liner is seen clearly in the hot flow compared with the cold flow. The values of $u'v'$ in the vortex downstream of the fuel jet is similar to those in the cold flow which are below 5 $m^2/s^2$.

The value of $u'v'$ at the point where the primary jet enters the flowfield is -40$m^2/s^2$. Further downstream of this jet flow, a local maximum of 91.5 $m^2/s^2$ is reached and this magnitude is higher than that in the cold flow. Similar to the primary jet of the cold flow, the shear stress upstream of the jet has a negative sign while the shear stress downstream has a positive sign. The part of the primary flow that enters the secondary zone in the upper part of the combustor has values of $u'v'$ slightly higher than those of the cold flow. In the secondary recirculation, the magnitudes of $u'v'$ are below 2.5 $m^2/s^2$. In the secondary zone, the top dilution jet enters with $u'v'$ at ~310 $m^2/s^2$ and decreases in magnitude as the jet penetrates across the flowfield. Just above the centerline in the secondary zone, $u'v'$ reaches a local minimum of -27.6 $m^2/s^2$ upstream of the dilution jet. This is at the location where
the primary jet velocity almost diminishes when it encounters the top dilution jet. Slightly downstream from this point is another local minimum in $\overline{u'v'}$ with a value of $-12.1 \text{ m}^2/\text{s}^2$ where the jet flow is seen turning drastically by about 90° and flowing towards the exit plane. In the lower part of this turning dilution jet flow, the values of $\overline{u'v'}$ are still higher than those in the cold flow. At the exit plane, $\overline{u'v'}$ increases from $3 \text{ m}^2/\text{s}^2$ near the top liner to a high of $25 \text{ m}^2/\text{s}^2$ and then decreases to a low of $-16 \text{ m}^2/\text{s}^2$ in the lower part of the flow before increasing back up to about $8 \text{ m}^2/\text{s}^2$ near the bottom liner. Overall, the magnitudes of $\overline{u'v'}$ are higher in the hot flow than in the cold flow.

**Hot Flow Section 2**

The contour and surface plots of the shear stress in Section 2 are shown in Figs. 123 and 124. In the primary vortex, $\overline{u'v'}$ is very low in magnitude over a large region in the upper front part of the combustor. Although the high shear stress area near the fuel injector is not found in this section, a local minimum of about $-4 \text{ m}^2/\text{s}^2$ is seen just upstream of the fuel inlet, which is still caused by the shearing effect of the fuel jet with the surrounding air flow. At the location where this occurs, the fuel jet and the primary jet are flowing towards the front-end against the flow in the primary vortex. Below this point is another local minimum with values of $-1.8 \text{ m}^2/\text{s}^2$, where the spread of the fuel jet enters into this plane. Slightly below this is a local maximum of $\overline{u'v'}$ in the fuel jet with magnitude of $56.1 \text{ m}^2/\text{s}^2$. Downstream along the fuel jet, $\overline{u'v'}$ remains fairly constant at about $30 \text{ m}^2/\text{s}^2$ all the way down to the bottom liner where the fuel jet strikes the liner. The vortex upstream of the fuel jet has lower magnitudes of $\overline{u'v'}$ than that in Section 1. In the vortex downstream of the fuel jet, a local minimum of $-6.1 \text{ m}^2/\text{s}^2$ is seen about this vortex center.

The shear stress in the primary jet behaves quite similar to that in Section 1 with a negative region upstream and a positive region downstream of the jet centerline. A local maximum of $54.9 \text{ m}^2/\text{s}^2$ is found in the primary jet near the bottom liner. Further downstream of the primary jet flow, the values of $\overline{u'v'}$ are higher than those in Section 1 since more primary flow enters the secondary flow in this plane. In the secondary recirculation, $\overline{u'v'}$ is rather low with gradients slightly higher than in the previous section due to the fact that a larger amount of mass flow above it causes the recirculation to lie closer to the bottom liner. In the secondary zone, the bottom dilution jet enters at about $-16 \text{ m}^2/\text{s}^2$. Upstream of this dilution jet just below the centerline, a local maximum of $18.2 \text{ m}^2/\text{s}^2$ is seen due to the strong influence of the top dilution jet from the other planes. Downstream of this is another local maximum with $\overline{u'v'}$ at $100.2 \text{ m}^2/\text{s}^2$, also due to the crossflow with the top dilution jet. Above the centerline there is a large negative $\overline{u'v'}$ region of the bottom dilution jet with a local minimum of $-55.8 \text{ m}^2/\text{s}^2$. At the exit plane, $\overline{u'v'}$ increases from near zero at the top liner to a high of about $30 \text{ m}^2/\text{s}^2$. It then decreases to about $-16 \text{ m}^2/\text{s}^2$ below the centerline, and increases back up to about $10 \text{ m}^2/\text{s}^2$ near the bottom liner.

**Hot Flow Section 3**

Figures 125 and 126 depict the contour and surface plots of $\overline{u'v'}$ in Section 3. Along the front dome liners, the shear stress is slightly above and below zero indicating that the near wall
flow is fairly smooth without much influence from the other flow features as in the previous sections. The zero contour passes through the center core of the primary vortex. Above the vortex center, $u'v'$ reaches a local maximum of 4 m$^2$/s$^2$; whereas below the vortex center, $u'v'$ reaches a local minimum of $-3.9$ m$^2$/s$^2$. Further downstream in the upper part of the primary flow, a local maximum of almost 10 m$^2$/s$^2$ is found at a location where the primary flow splits with some recirculating back to the front-end and some going downstream. In the primary jet near the bottom liner, there are still regions with the negative and positive $u'v'$ upstream and downstream of it. A local maximum of 10.5 m$^2$/s$^2$ is observed and this is much lower in magnitude than those found in the other two sections. The region of influence of this primary jet has significantly decreased in the shear stress contour plot. In the core of the primary flow below the thumb-nail louvre, a local maximum of 17.6 m$^2$/s$^2$ is generated since $u'v'$ decreases towards the top liner, the secondary recirculation and the dilution jet downstream.

In the secondary recirculation, the shear stress decreases from 4 m$^2$/s$^2$ on the outer edge to $-3$ m$^2$/s$^2$. In the secondary zone, the bottom dilution jet enters at $\sim 7$ m$^2$/s$^2$ and reaches a local minimum of $-57$ m$^2$/s$^2$ upstream and a local maximum of 98 m$^2$/s$^2$ downstream of the center of the dilution jet. The negative and positive regions of $u'v'$ are much more clearly defined here than in Section 2 since this section is at the center of the bottom dilution hole whereas Section 2 is off-centered. Near the top liner in the secondary zone, the more horizontal outflow of the primary flow causes $u'v'$ to become positive again and the magnitude of the shear stress there is below 10 m$^2$/s$^2$. At the exit plane, $u'v'$ increases from 5 m$^2$/s$^2$ to a high 19 m$^2$/s$^2$ above the centerline of the combustor. Then it decreases to $-16$ m$^2$/s$^2$ in the wake of the dilution jet, and increases back up to about 3 m$^2$/s$^2$ near the bottom liner.

### 6.4.4 Shear Stress Intensity SI

*Cold Flow Section 1 (Fuel Injector Plane)*

The shear stress intensity in Section 1 of the cold flow is shown in Fig. 127. In the primary vortex, the shear intensity is low in front of the ignitor because of the higher velocity magnitude and the lower shear stress in this region. Towards the center of the primary vortex, because of the decrease in velocity, SI increases and a local maximum of about 2.0 is seen. However, along the top and front dome liners, since both the shear stress and the velocity are higher than those in the core of the primary vortex, SI becomes lower. Upstream near the fuel injector inlet, the shear stress and the velocity increase due to the increasing entrainment of the fuel jet with the surrounding air. Hence, SI remains low at about 0.2. Along the fuel jet trajectory, SI is fairly constant and ranges between 0.2 and 0.4 till near the bottom liner. The shear stresses are higher on the downstream edge of the fuel jet than those on the upstream. Also, where the shear stress is a local maximum, SI is also a local maximum at 0.78. In the vortical flow upstream of the fuel jet at the bottom dome area, SI is as high as 2.0 since the shear stress is high, and the velocity is rather low and fairly constant there. Similarly, the vortex downstream of the fuel jet also has a local maximum of 2.0 in SI. Just below the centerline of the combustor in between the fuel jet roll-up vortex and the primary jet, a local minimum of near zero is found.
The value of SI at the primary jet inlet is about 0.2. At the shear layer slightly above the inlet, a local maximum of 1.0 is seen. Further downstream in the core of the primary jet flow, a local minimum of 0.04 is found above the centerline where the shear stress is small and the velocity is high. In the secondary recirculation, a local maximum of 3.6 is reached at a point where the velocity is a local minimum, which is the center of this vortical flow. Towards the bottom liner in this vortex, SI decreases to about 0.2 since the velocity increases and the shear stress decreases near the bottom liner. The primary flow enters the secondary zone in the upper part of the combustor with SI at about 0.2. In the secondary zone, the top dilution jet enters at SI at about 0.3 which is close to that in the fuel jet. In the upstream shear layer, SI increases slightly to a local maximum of 1.36. At the point where the jet flow turns rapidly downstream, SI reaches a local maximum of 1.23 due to the low velocity there. At the exit plane, the values of SI ranges between 0.2 and 0.4.

Hot Flow Section 1 (Fuel Injector Plane)

The shear stress intensity in Section 1 of the combusting flow is shown in Fig. 128. In the primary vortex, the values of SI are generally higher than those seen in the cold flow but a larger region of SI at 0.2 is found in front of the ignitor. A local maximum of 6.56 is found on the upstream of the fuel jet, which is three times as high as the local maximum value in the primary vortex of the cold flow. The shear intensity SI at the fuel jet inlet is below 0.2 which is slightly lower than the level found in the cold flow. The shear intensity is reduced with combustion near the fuel injector inlet because of the local laminarization due to the increase in laminar viscosity with temperature. As the fuel jet cuts across the primary zone, the shear stress intensity in the core of the fuel jet remains at below 0.2 until it gets close to splashing onto the bottom liner. The shear stress intensity contours are seen much more evenly distributed on either side of the fuel jet axis than in the cold flow case. The small vortex upstream of the fuel jet near the bottom of the front dome has much higher values of SI with combustion. The local maximum there reaches above 2.0. Downstream of the fuel jet, the vortex has a local maximum of 1.0 since the velocity there has decreased to near zero.

The primary jet has SI of about 0.2 at the jet inlet and the intensity remains fairly constant around 0.2 before it drops to a local minimum of near zero just above the combustor centerline as the jet flows up towards the top liner. This local minimum occurs at a point on the axis of the primary jet where the velocity is high and the shear stress is near zero. In the secondary recirculation, the shear intensity is below 0.2 similar to the cold flow case. In the secondary zone near the inlet of the top dilution jet, SI is about 0.3. The local maximum attained upstream of this dilution jet is nearly the same as that found in the cold flow. However, the next local maximum upstream of this jet is 3.1 compared with 1.0 in the cold flow. This is due to the high shear stress found in the crossflow of the hot expanding primary flow with the cold dilution jet. Then at the point where the dilution jet rolls rapidly towards the exit plane, a local maximum of SI at 2.0 is found due to a rapid drop in the velocity magnitude. The remaining part of the dilution jet flow in the lower part of the combustor has values of SI similar to those found in the cold flow. At the exit plane, values of SI range between 0.2 and 0.4, which do not differ much from those of the cold flow.
**Hot Flow Section 2**

The shear stress intensity in Section 2 are shown in Fig. 129. In the primary vortex, because of the low velocity in the center, SI becomes a local maximum at 2.1. Moving away from the vortex center, SI decreases to below 0.2 in front of the ignitor. However, beneath the louvre in the top liner, due to the increase in shearing action between the incoming louvre flow and the recirculating primary flow, SI is high. In front of the fuel injector inlet, SI is about 1.0, which is higher than that in Section 1. At the point where the shear stress has a local minimum of near zero approximately two inlet diameter away from the fuel jet inlet, SI is also a local minimum around 0.2. As the fuel jet penetrates into this plane, SI increases to and remains at about 0.4 along the axis of the fuel jet. Below the centerline, SI decreases to below 0.2. Then SI increases to near 0.4 as the fuel jet impinges onto the bottom liner. One local maximum is found on each side of the fuel jet about the centerline of the combustor. At the upstream edge of the fuel jet where the velocity there is fairly low, the shear intensity reaches a peak at 1.8. On the downstream edge of the fuel jet, SI reaches a peak of about 1.0. In the vortex upstream of the fuel jet, the shear stress intensity at the center is 2.0. In the vortex downstream of the fuel jet, SI is mostly about 0.4 with a local maximum value of 1.0.

The shear stress intensities of the primary jet are quite similar to those in Section 1. The primary flow that enters the secondary zone has SI values around 0.4. In the core of the primary jet above the combustor centerline, SI has decreased to almost zero. In the secondary recirculation, SI has a local maximum of 2.3 about the vortex center in which the velocity is also a local minimum. Outside the core region of this vortex, SI varies from about 0.4 to below 0.2 near the bottom liner. At the downstream edge of the secondary recirculation, the shear stress is near zero and SI is low at around 0.2. In the secondary zone near the inlet of the bottom dilution jet, SI is 0.2 and this intensity level remains fairly constant for a few inlet diameter downstream before it increases further. Upstream of this dilution jet above the centerline of the combustor, SI has a local maximum of almost 6.5 due to the opposing flow between the top and bottom dilution jets and the crossflow of these jets with the oncoming primary flow. The velocity here is a local minimum. As the primary flow leaves along the top liner, SI gradually decreases to about 0.2 near the top. At the exit plane, SI remains fairly constant at 0.2.

**Hot Flow Section 3**

Figures 130 shows the shear stress intensity of the hot flow in Section 3. Along the front dome liners, the shear stress intensity is below 0.1, which is very much lower than those found in the other two sections investigated. In the primary vortex, two local maximums are observed, one above and one below the vortex center. Since the contours of the shear stress and the velocity magnitudes above are quite similar to those below the vortex center, except for being compressed and slightly distorted, the values of these two peaks are not too different. The one above has a SI slightly over 1.0 and the one below is slightly under 1.0. The shear intensity at the center of the primary vortex is about 0.4. Just downstream of the vortex center, a local minimum of SI near zero is found at a point where the shear stress is zero and

103
the velocity is increasing. The bulk of the primary flow in this zone has SI below 0.4. In the upper part of the combustor in the vicinity of the fuel injector, the shear stress increases but the velocity decreases due to the splitting of the primary flow. Hence a high of about 1.5 is seen upstream of the fuel injector location.

In the primary jet flow away from the jet inlet, a local minimum of zero in SI is observed upstream since the shear stress changes sign there. Downstream of this jet, SI increases at the edge of the secondary recirculation. At about the center of the secondary recirculation, SI increases to a local maximum of about 1.5. Just underneath the thumb-nail louvre, SI increases from 0.2 in the primary flow to about 0.6 near the top liner. This is because the expansion of the hot combusting gas splashes and flows out of this plane in the tangential direction. In the secondary zone at the inlet of the bottom dilution jet, SI is at 0.1. As it penetrates across the flowfield, SI increases to a peak of 1.16 upstream and only to about 0.4 downstream of the jet. In the upper part of the combustor where the primary flow mixes with the dilution jet, SI decreases towards the top liner and a local minimum of 0.02 is seen. At the exit, SI varies from 0.1 in the upper part to 0.4 in the lower part of the exit plane.

6.4.5 Reynolds Stress Correlation Coefficient $C_{uv}$

*Cold Flow Section 1 (Fuel Injector Plane)*

The Reynolds stress correlation coefficient contour plot for the cold flow in Section 1 is shown in Fig. 131. The contour profile of $C_{uv}$ is quite similar to the shear stress. Upstream of the fuel jet are two regions. One is at the verge of the shear layer of the fuel jet near the fuel injector with the region elongated in the longitudinal direction. The correlation coefficient ranges from -0.3 to 0.2, with the higher value towards the top liner and the fuel jet. The second one is located in front of the ignitor along the front liner. The coefficient in this region is very similar to the previous one with $C_{uv}$ which varies from about -0.1 to 0.2. The shape of this region is elongated in the transverse direction. Note that although the shear stress in the shear layer of the fuel jet near the fuel injector is much higher than that in the primary vortex, the correction coefficient of these two regions are not very different. Under the ignitor near the front-dome liner, $C_{uv}$ increases to 0.4. $C_{uv}$ of the fuel jet at the inlet is about 0.2; upstream, $C_{uv}$ decreases, whereas downstream, $C_{uv}$ increases. Further downstream of this fuel jet, $C_{uv}$ increases to about 0.3 along the jet centerline. In the vortex upstream of the fuel jet, $C_{uv}$ increases to 0.5. On the contrary, the vortex downstream of the fuel jet has very low $C_{uv}$ at 0.1. At the bottom louvre just downstream of where the fuel jet strikes the liner, $C_{uv}$ increases from 0.3 to 0.7.

At the inlet of the primary jet, $C_{uv}$ is -0.3. On either side of this jet inlet, an increase in $C_{uv}$ is seen. Further downstream of the jet flow, $C_{uv}$ reaches a local minimum of -0.31. Along the shear layer downstream of the primary jet, $C_{uv}$ increases to a local maximum of about 0.5. Above the secondary recirculation zone, $C_{uv}$ in the primary flow is about 0.3. Within the secondary recirculation, the correlation coefficient decreases from 0.3 to -0.2 near the bottom liner. In the secondary zone near the inlet of the top jet, $C_{uv} \sim 0.7$. It is seen that $C_{uv}$
decreases as it penetrates into the combustor to a low of $-0.1$ where the flow turns rapidly towards the exit. Along the upstream shear layer, $C_{uv}$ decreases to about $-0.1$. In the lower part of the top jet flow, $C_{uv}$ increase back up to a local maximum of $0.3$. At the upper half of the exit plane, $C_{uv}$ increases as the flow exits. Opposite to this trend is the flow in the lower part of the combustor where $C_{uv}$ decreases in the wake flow.

**Hot Flow Section 1 (Fuel Injector Plane)**

The contours of the Reynolds stress correlation coefficient in Section 1 of the hot flow is shown in Fig. 132. In the primary zone, the narrow strip of recirculating flow along the front dome liner below the ignitor has higher $C_{uv}$ than in the cold flow. In the primary vortex near the ignitor, the values of $C_{uv}$ do not differ appreciably from those in the cold flow but this region is forced upward by the stronger upstream vortex of the fuel jet. Beneath the top liner and upstream of the fuel injector is another shear region. Near the top liner, $C_{uv}$ is also higher with the hot flow. However, when approaching the shear layer of the fuel jet upstream of the fuel injector inlet, $C_{uv}$ decreases to a local minimum of about $-0.2$. Also in the core of the fuel jet near the inlet, $C_{uv}$ is slightly lower than that in the cold flow. In the shear layer downstream of the fuel injector inlet, $C_{uv}$ is also lower in the hot flow. Hence, the correlation coefficient around the fuel injector is lower in the hot flow when compared with the cold flow. This is because of the decrease in the shear stress due to the local laminarization of the jet near its inlet as the temperature of the flow increases with combustion. Similar to the fuel jet in the cold flow, $C_{uv}$ is quite constant at about $0.2$ in the centerline of the fuel jet. However, higher values of $C_{uv}$ in the shear layers are found with the hot flow. Two local maxima of $C_{uv}$ are seen on each side of the shear layer of the fuel jet at about $0.4$, several inlet diameter away from the fuel injector. Near the bottom liner where the fuel jet strikes, $C_{uv}$ increases rapidly from about $0.2$ to over $0.5$. $C_{uv}$ in the vortex upstream of the fuel jet is higher than in the cold flow although the local maximum in both cases are $0.5$. In the vortex downstream of the fuel jet, $C_{uv}$ decreases in value rapidly in the shear layer to a low of about $-0.1$ in the center. Then upstream of the primary jet, $C_{uv}$ increases to a local maximum of $0.4$, which is much higher than in the cold flow.

The primary jet enters with $C_{uv}$ at $-0.5$. To the left of the primary jet’s centerline, $C_{uv}$ is negative and reaches a local maximum of $-0.42$ at the centerline of the combustor. To the right of the primary jet’s centerline, $C_{uv}$ is positive and has a local maximum of $0.51$ just below the centerline of the combustor. In the secondary recirculation, $C_{uv}$ ranges from about $0.2$ above the vortex center to about $-0.2$ near the bottom liner. In the secondary zone, the top dilution jet enters with $C_{uv}$ above $0.5$ and as it penetrates across the flowfield, $C_{uv}$ decreases to about zero at the centerline where the jet flow turns drastically downstream. Below this point, $C_{uv}$ increases up to $0.3$ before it decreases down to zero rapidly near the bottom liner. Upstream of the top dilution jet is the crossflow with the primary flow. $C_{uv}$ are negative here and the magnitudes are much larger than in the cold flow. A local minimum of about $-0.4$ is located where the primary flow gets strongly influenced by the top dilution jet and the shear stress also has a local minimum in this vicinity. At the exit plane, $C_{uv}$ varies between $-0.2$ and $0.2$. 

105
Hot Flow Section 2

Figure 133 depicts the contour plot of the Reynolds stress correlation coefficient in Section 2. The two distinct regions upstream of the fuel jet seem to have merged. The primary vortex in the upper front of the combustor has a local minimum of $-0.3$ in front of the ignitor. At the center of the vortex, $C_{uv}$ is about 0.25. Towards the top liner, $C_{uv}$ increases to above 0.5. Near the fuel injector inlet, $C_{uv}$ is higher here than in Section 1 because the flow from the primary jet turns upstream and the shear stress increases near a solid boundary which is the fuel injector here. The correlation coefficient of the fuel jet is 0.2 in the jet's centerline. It then increases slightly to 0.3 when the jet strikes the bottom liner. In the vortex upstream of the splashing, $C_{uv}$ is lower than in Section 1. The values of $C_{uv}$ found in the vortex downstream of the fuel jet are very similar to those in the last section, and the local minimum is about the same at $-0.1$.

$C_{uv}$ in the primary jet is similar to that in Section 1 with a negative region to the left and a positive region to the right of the jet's centerline. However, the negative region is now smaller and the values of $C_{uv}$ are smaller. The local minimum is about $-0.3$ instead of $-0.4$ as in Section 1. In the positive region, $C_{uv}$ is very similar to those in Section 1 and the local maximum is identical at 0.5 except that the location is shifted downstream and lower. In the flow downstream of the fuel injector dome, $C_{uv}$ is also high at about 0.5 as observed in Section 1. The secondary recirculation zone also has values of $C_{uv}$ similar to those in Section 1 except that the positive $C_{uv}$ region extends further down closer to the bottom liner. In the secondary zone, the bottom dilution jet enters with $C_{uv}$ at about $-0.2$ and the value of $C_{uv}$ increase slightly to zero near the centerline and then decrease gradually as it cuts across the secondary zone towards the top liner. Seen upstream of this flow is the strong interaction of this bottom dilution jet with the top dilution jet causing very high shear stress there as a result. Hence, a local minimum of $-0.43$ in $C_{uv}$ is also present here. In the wake of the bottom dilution jet, $C_{uv}$ has a local maximum of about 0.5 due to the high shearing action between the more horizontal flow going towards the exit as seen in Section 1 and the near vertical dilution jet going towards the top liner. At the exit plane, $C_{uv}$ also varies between $-0.2$ and 0.2 as in Section 1, except near the bottom liner where $C_{uv}$ increases rapidly to above 0.5.

Hot Flow Section 3

Figure 134 shows the Reynolds stress correlation coefficient in Section 3. Near the front dome, the zero contour of $C_{uv}$ goes through the center of the primary vortex. Above this vortex center, $C_{uv}$ is positive and increases to a local maximum of about 0.3. Below the vortex center, $C_{uv}$ is negative and decreases to a local minimum of about $-0.3$ which is like an equal counterpart of the local maximum above it. The negative region of $C_{uv}$ is larger than the positive region in the transverse direction because the primary vortex center is strongly constrained to the front-top of the combustor by the fuel jet in the fuel injector plane. Approximately in the middle of the primary zone, a local maximum of 0.25 is found in the primary flow. Above this and slightly downstream, the primary flow splits with part going upstream and part continuing downstream. A local maximum of about 0.4 in $C_{uv}$ is
found here. Near the inlet of the primary jet, $C_{uv}$ is low about $-0.3$ and is increasing as the primary jet flows upward. The negative $C_{uv}$ region reaches a high of $-0.1$ and the positive $C_{uv}$ region reaches a high of $0.2$ at the shear layer. It is surprising to find out that the orientation of these $C_{uv}$ contours in the primary jet do not agree with the velocity vectors in this section. Rather, it matches with the orientation of the velocity vectors of the primary jets near the inlet in both Section 1 and 2. This indicates that the profile of the shear stress still preserves the inlet jet flow orientation in planes that are away from the inlet.

In the region of the primary flow under the thumb-nail louvre, $C_{uv}$ is relatively higher than the others in this plane. A maximum of $0.56$ is reached in this region above the secondary recirculation. In the secondary recirculation zone, $C_{uv}$ varies from $-0.2$ to $0.2$. In the secondary zone, the bottom jet enters the combustor with $C_{uv}$ varying from $-0.1$ to $0.2$. A high gradient of $C_{uv}$ exists upstream of this jet in the shear layer. As in the primary jet, a negative and a positive region is found to the left and right of the bottom dilution jet’s centerline, respectively. A local minimum of $-0.36$ is reached in the negative region and a peak of about $0.4$ is found in the positive region which is lower than the minimum point. The negative region is pushed downstream in the upper half of the dilution jet flow due to the strong crossflow with the oncoming flow. Near the top liner, $C_{uv}$ increases to $0.2$ in the secondary zone. At the exit plane, $C_{uv}$ is fairly constant at $0.1$ and then the value of $C_{uv}$ decreases to a low of $-0.2$ in the lower part of the exit before it increases back up to about $0.1$ near the bottom liner.

### 6.4.6 Ratio of the Normal Stress Intensities

*Cold Flow Section 1 (Fuel Injector Plane)*

The ratio of the normal stress intensities $\sqrt{\frac{u'^2}{v'^2}}$ for the cold flow in Section 1 is shown in Fig. 135. Above the centerline in the front dome, the ratio is between $0.8$ and $1.0$. However, further downstream into the shear layer of the fuel jet, the ratio increases and a region upstream of the fuel injector with a high ratio of $1.75$ is seen and a local maximum of $1.78$ below it is found. At the fuel jet inlet, the ratio is about $0.6$. Then as the fuel jet cuts across the primary zone, the ratio increases to unity and remains fairly constant along the centerline of the fuel jet. To the left of the jet centerline, the ratio is above unity and to the right of it, the ratio is below unity. At the bottom liner where the fuel jet splashes, the ratio increases slightly to about $1.25$. The small vortex upstream has a ratio of about $1.0$. In the vortical flow downstream, higher ratios are found near the bottom liner and they decrease to about $0.8$ at the center and then continue to decrease into the fuel jet.

The primary jet has a very low ratio of $0.5$ at the inlet and the ratio increases to $0.9$ before decreasing gradually to a local minimum of $0.57$ under the fuel injector above the centerline. In the secondary recirculation, the ratio is $1.25$ at the outer edge of the recirculation and it drops to about $0.7$ near the bottom liner because of the decrease in $u'^2$. The primary flow that leaves the primary zone above the secondary recirculation zone has values of the ratio between $0.7$ and $1.0$. In the core of the top dilution jet, the ratio is $0.5$. Higher values and gradients of ratio exist upstream of the shear layer. A local maximum of $1.6$ is seen just above
the centerline where the primary flow splits and the top dilution jet starts having influences
on the secondary recirculation. At the center of the dilution jet, the ratio decreases slightly to
a local minimum of 0.44 where the dilution jet turns downstream rapidly. Then in the lower
part of the secondary zone, the ratio increases up to 1.0 near the bottom liner. At the exit, the
ratio is approximately 0.6 to 0.7 over the most part and an increase of up to 1.0 is observed
only near the bottom liner.

**Hot Flow Section 1 (Fuel Injector Plane)**

Figure 136 shows the ratio of the normal stresses intensities in Section 1 of the hot flow.
In the front dome near the ignitor, the ratio is lower in the hot flow than in the cold flow
with values ranging from 0.6 to 1.0. Upstream of the fuel jet, very high ratio are found in the
shearing flow between the primary vortex and the expanding fuel jet. A local maximum of
about 2.0 is observed near the fuel injector inlet and another one further downstream. The
fuel jet has an inlet ratio of about 0.6 as in the cold flow. Further downstream along the
centerline of the fuel jet, the ratio increases to and remains fairly constant at 1.0. Upstream
of the jet’s centerline, the ratios are above 1.0; downstream of the jet’s centerline, the ratios
are below 1.0. A local maximum of 1.8 is attained at the upstream shear layer where the
upstream vortex recirculates back into the fuel jet about the centerline. Downstream of the
fuel jet, there is a local minimum of 0.53 about the vortex center, and another local minimum
of 0.45 is observed above the centerline where the vortical flow is being entrained back into
the fuel jet.

At the primary jet inlet, the ratio is 0.8 and it increases rapidly to 1.0 near the inlet
before decreasing in value as it penetrates into the flowfield. At the upstream edge of the
secondary recirculation, a local maximum ratio is seen. The ratio is decreasing from a value
of about 1.25 at the outer edge of the recirculation to about 0.8 near the bottom liner. In the
secondary zone, the top dilution jet is entering with a ratio of about 0.6. In this dilution jet,
the ratio decreases very slowly to a local minimum of 0.4 above the centerline of the combustor
where the dilution jet changes the flow direction rapidly downstream. The ratio then increases
slightly and gradually up to 0.6 near the bottom liner in the lower part of the dilution jet flow.
Due to the crossflow with the oncoming primary flow, $u'^2$ is higher on this jet’s upstream and
so are the values and gradients of the ratio. There is a local maximum of 1.6 just above the
centerline of the combustor upstream of the dilution jet where the primary flow splits at the
downstream edge of the secondary recirculation zone, and the flow there is being influenced
by the top dilution jet. At the exit plane, the ratio increases from 0.6 near the top liner to 0.9
near the bottom liner.

**Hot Flow Section 2**

Figure 137 depicts the ratio of the normal stresses intensities in Section 2. In the primary
vortex, the ratio is 1.0 in the vortex center. Below this center, the ratio decreases to a local
minimum of 0.5 at the lower edge of the primary vortex. Although the fuel jet does not actually
appear until several fuel injector inlet diameter downstream of the fuel jet’s centerline, the
closely-spaced contour lines do originate just upstream of the fuel injector similar to those in the hot and cold flows of Section 1. Upstream where the fuel jet appears, the ratio reaches a maximum of 0.2. Then at the centerline where the upstream vortical flow is recirculating back into the fuel jet, a local maximum of 1.9 in the ratio is seen. The ratio along the centerline of the fuel jet is like those in Section 1 of the hot and cold flows at 1.0. In the vortex downstream of the fuel jet, a local minimum of 0.57 is found above and slightly downstream of the vortex center.

The core of the primary jet has a ratio of 0.7 up to about the centerline of the combustor before it decreases as it flows upward and reaches a local minimum of 0.46. Downstream of the primary jet within the secondary recirculation, the ratio varies from a local maximum of 1.5 near the primary jet to a low of 0.8 near the bottom liner. At the upper edge of the secondary recirculation zone where the primary flow splits up, the ratio has a local maximum of 1.6. The bottom dilution jet has a low ratio of 0.4 at its inlet. The ratio increases slightly to 0.5 in the core of the jet before it decreases to a local minimum of 0.38 at the centerline. As the primary flow enters and leaves the secondary zone mainly near the top liner, the ratio decreases from 1.0 to 0.7 at the upper part of the exit.

**Hot Flow Section 3**

The contours of the ratio of the normal stress intensities in Section 3 are shown in Fig. 138. In the primary zone near the front dome, the ratio is less than 1.0 except near the top and bottom liners. A local minimum of 0.6 is located slightly below the primary vortex center. Further downstream of the primary vortex center, a local maximum of 1.2 is reached because \( \overline{u'^2} \) is increasing whereas \( \overline{v'^2} \) is decreasing in this area. At about the fuel injector location, the hot combusting mixture reaches the top liner and flows across this section in the tangential direction. The ratio is above 1.0 in this region. Near the bottom liner in the primary zone just upstream of the primary jet, the ratio reaches a high of about 1.4.

The core of the primary jet has ratio of about 0.7. Higher ratios are found on either side of the shear layer of this primary jet. Above the centerline of the combustor, a local minimum of 0.53 is observed in the primary jet flow. The ratio in the secondary recirculation zone of this section ranges from 1.0 to 1.4, which are higher than those in the other sections. Due to difficulties in measuring \( \overline{v'^2} \) near the liners, the ratio at the bottom dilution inlet is not certain. The core of the dilution jet further downstream is about 0.5 to 0.6. Higher gradients and values of the ratio are found upstream due to the crossflow with the primary jet. The primary flow leaves the secondary zone along the top liner with a ratio of 0.8. At the exit plane, the ratio is almost at 0.9 right across.

### 6.4.7 Turbulence Kinetic Energy \( k \)

**Cold Flow Section 1 (Fuel Injector Plane)**

The contour and surface plots of the turbulence kinetic energy \( k \) in Section 1 of the cold flow are shown in Figs. 139 and 140. In the primary vortex, the turbulence kinetic energy is
low at 5 \((\text{m/s})^2\) around the top of the front dome and it increases in value towards the fuel jet. At the fuel jet inlet, \(k\) is 1000 \((\text{m/s})^2\). It then decays rapidly along the centerline of the fuel jet initially. About two inlet diameter downstream, the decay is more gradual and when the fuel jet impacts at the bottom liner, \(k\) has dropped to about 70 \((\text{m/s})^2\). The gradient of \(k\) downstream is higher than upstream of the fuel jet. The vortex upstream of the fuel jet at the bottom liner has much higher values of \(k\) than the vortex downstream of the fuel jet.

The primary jet has a core region in which \(k \sim 150 \text{ (m/s)}^2\). As this jet penetrates the primary zone and flows upward towards the fuel injector, the turbulence kinetic energy decays down to about 20 \((\text{m/s})^2\) in the upper part of the primary flow. In the secondary recirculation, \(k\) decreases from about 20 \((\text{m/s})^2\) at the upper edge of this zone to a low of about 5 \((\text{m/s})^2\) near the downstream shear layer of the primary jet. The primary flow enters the upper part of the secondary zone with \(k = 15 \text{ (m/s)}^2\). In the secondary zone, the top dilution jet enters with \(k = 300 \text{ (m/s)}^2\). The turbulence kinetic energy decreases slightly and then increases very gradually to a maximum of 349 \((\text{m/s})^2\) at the centerline where the top dilution jet rotates rapidly towards the exit. Then further down towards the bottom liner, \(k\) decreases gradually to 30 \((\text{m/s})^2\). A higher gradient of \(k\) is found upstream of this dilution jet because of the crossflow with the primary jet. At the exit plane, \(k\) increases from about 30 \((\text{m/s})^2\) at the top to about 70 \((\text{m/s})^2\) near the centerline of the combustor and then decreases back to 30 \((\text{m/s})^2\) at the bottom.

**Hot Flow Section 1 (Fuel Injector Plane)**

The contour and surface plots of the turbulence kinetic energy \(k\) in Section 1 of the hot flow are shown in Figs. 141 and 142. The primary vortex has higher turbulence kinetic energy with combustion. The value of \(k\) near the fuel jet inlet is about 800 \((\text{m/s})^2\). In slightly over one fuel injector inlet diameter downstream along the fuel jet, the turbulence kinetic energy in the hot flow is already higher than that in the cold flow. The values of \(k\) near the fuel inlet are lower than in the cold flow because of the local laminarization of the flow due to the increase in laminar viscosity with temperature of combustion. However, further downstream in the fuel jet, \(k\) has a slower decay rate than in the cold flow. At the point where this fuel jet strikes the bottom liner, the value of \(k\) is about 100 \((\text{m/s})^2\) compared with 70 \((\text{m/s})^2\) in the cold flow. The region engulfed by the closely packed contours is larger with the hot flow because of the expansion of the fuel jet with combustion. The two vortices upstream of the fuel jet have very much higher values of \(k\) in the hot flow. The vortex downstream of the fuel jet has lower values of \(k\) than those upstream, but still they are higher than in the cold flow.

The core region of the primary jet has a value of \(k\) slightly above 100 \((\text{m/s})^2\) which is lower than that in the cold flow. But the decay rate of \(k\) in this primary jet, as it penetrates into the flowfield, is lower than that in the cold flow. Several inlet diameter downstream of this primary jet, the value of \(k\) is already higher than in the cold flow. A local minimum of 12.4 \((\text{m/s})^2\) is found slightly downstream of the fuel jet's shear layer where the primary flow is entrained into the strong fuel jet above the centerline of the combustor. In the secondary recirculation zone, the turbulence kinetic energy is similar to that in the cold flow except that the gradient is more evenly distributed in the longitudinal direction. The magnitude
of turbulence kinetic energy in the primary flow that enters the secondary zone above the secondary recirculation is about 30 \( (m/s)^2 \) which is also higher than in the cold flow. In the secondary zone, the top dilution jet enters the combustor with \( k \) at about 300 \( (m/s)^2 \) and increases in magnitude as it cuts across the flowfield, reaching a local maximum of 494 \( (m/s)^2 \) just above the centerline of the combustor due to the strong influence of the bottom dilution jet in the other planes. In the lower part of the dilution jet flow, \( k \) decays gradually to about 60 \( (m/s)^2 \) near the bottom liner. At the exit plane, the value of \( k \) increases from 60 \( (m/s)^2 \) at the top to 100 \( (m/s)^2 \) and then decreases down to below 40 \( (m/s)^2 \) near the bottom liner.

Hot Flow Section 2

Figures 143 and 144 are the contour and surface plots of the turbulence kinetic energy in Section 2. In the primary vortex, the values of the turbulence kinetic energy is lower in this plane than in the previous one. A high gradient of \( k \) upstream of the fuel injector inlet is still found in this plane but is now lower than in Section 1. In front of the fuel injector inlet, \( k \) is at about 60 \( (m/s)^2 \). Then further downstream of the fuel jet flow, \( k \) increases in value to a local maximum of 213 \( (m/s)^2 \) approximately where the strong fuel jet intrudes into this plane. The values of \( k \) from this local maximum point to the bottom liner are slightly lower than those seen in Section 1. Also, the values of \( k \) at the shear layers are much lower in this plane. In the upstream vortex near the bottom liner, \( k \) is also lower. However, in the vortex downstream of the fuel jet, \( k \) is very similar to that in Section 1 except that this region is extended slightly downstream due to the expansion effect of combustion.

The peak value of \( k \) near the primary jet inlet is 175 \( (m/s)^2 \). It then decreases as it penetrates into the flowfield. The primary flow that enters the secondary zone in the upper half of the combustor above the secondary recirculation has values of \( k \) varying between 20 and 30 \( (m/s)^2 \). The turbulence kinetic energy within the secondary recirculation zone is only slightly lower than that in Section 1. In the secondary zone, the bottom dilution jet enters with \( k \) at about 200 \( (m/s)^2 \). In the core of this dilution jet, \( k \) increases to a maximum of 497 \( (m/s)^2 \) just below the centerline of the combustor due partially to the turning effect of the top dilution jet in Section 1. This maximum is almost identical to the one in the fuel injector plane inside the top dilution jet which is located at the centerline of the combustor. In the upper half of the combustor, as the primary flow mixes with the bottom dilution jet near the top liner, \( k \) continues to decrease. At the exit, \( k \) is \( \sim 100 \ (m/s)^2 \) in the upper half and it decreases gradually in the lower half to \( \sim 30 \ (m/s)^2 \) near the bottom liner.

Hot Flow Section 3

The contour and surface plots of \( k \) in Section 3 are shown in Figs. 145 and 146. Very low values of turbulence kinetic energy at below 4 \( (m/s)^2 \) are found along the front dome and bottom liners. In the primary vortex, the values of \( k \) are appreciably smaller than those observed in the last two sections. At about the center of the primary vortex, \( k \) is high at about 14 \( (m/s)^2 \). A local maximum of 14.6 \( (m/s)^2 \) is seen below the centerline and above the bottom louvre. The primary jet has a local maximum of 80 \( (m/s)^2 \) near the bottom liner.
and a high gradient of $k$ on its upstream. The secondary recirculation zone has higher values of $k$ here than in the previous two sections. The primary flow around the fuel injector and further downstream near the top liner has values of $k$ about 20 (m/s)$^2$. The primary flow that enters the secondary zone in the upper part of the combustor has values of $k$ similar to those found in the other sections. In the secondary zone, the bottom dilution jet enters with $k \sim 100$ (m/s)$^2$ as it penetrates across the flowfield, reaching a high of 300 (m/s)$^2$ about the centerline. The maximum value found in this zone is less than those present in the other two sections indicating that the interactions between the top and bottom dilution jets have decreased because of the higher mass of outflow from the primary zone in this section. Because of the same reason, near the top liner the values of $k$ are lower than those in Section 2. At the exit plane, the values of $k$ are lower near the top and bottom liners than those in the other two sections, but they are quite similar in the center part of the exit.

6.4.8 Ratio of $\left| \overline{u'v'} \right| / k$

_Cold Flow Section 1 (Fuel Injector Plane)_

The ratio of the shear stress $\overline{u'v'}$ and the turbulence kinetic energy $k$ in Section 1 of the cold flow is shown in Fig. 147. This ratio provides information on the amount of momentum transfer due to turbulence motion in comparison with the amount of mixing due to turbulence. Upstream of the fuel jet along the top liner and in the primary vortex, the ratio is basically below 0.2. At the fuel jet inlet and along the centerline of the jet downstream, the ratio of the shear stress and the turbulence kinetic energy is fairly constant at about 0.2. Near the bottom liner where the fuel jet splashes, the ratio increases and a local maximum of about 0.5 is found upstream of the shear layer where the incoming air from the front louvre opposes the splashing fuel jet. An increase in the ratio to about 0.3 is seen about the centerline on the downstream of the fuel jet shear layer. Much higher ratios between 0.3 to 0.5 are found downstream than upstream of the fuel jet inlet. The ratio in the vortex downstream of the fuel jet is very low. Around the bottom louvre, the ratio increases rapidly.

The ratio in the core of the primary jet is 0.1. As it penetrates the flowfield, the ratio decreases very slowly to a local minimum of 0.02 before increasing gradually up to about 0.5 near the dome of the fuel injector. The highest ratio in its upstream shear layer is only about 0.2, whereas the ratio increases to a local maximum of near 0.5 in the shear layer downstream. In the secondary recirculation, the ratio decreases from the outer edge from 0.4 to 0.1 in the lower part of the recirculating flow. In the secondary zone, the ratio is high at 0.6 near the top dilution jet inlet. It then decreases to 0.1 as it cuts across the flowfield. A local minimum of near zero is observed upstream of the dilution jet where the primary flow is encountering the strong dilution jet. In the lower part of the dilution jet flow, the ratio reaches a local maximum of 0.3.
**Hot Flow Section 1 (Fuel Injector Plane)**

Figure 148 depicts the ratio of $\overline{uv}$ and $k$ in Section 1 of the hot flow. The ratios in the primary vortex and upstream of the fuel injector inlet are similar to those in the cold flow. At the fuel jet inlet, the ratio is slightly lower in the hot flow. Further downstream in the jet, the ratio increases to about 0.2 and remains fairly consistent at this level in the core; near the bottom where it strikes the liner, the ratio increases rapidly to about 0.6. The ratio in the shear layer upstream and downstream of the fuel jet reaches 0.3. A local maximum of 0.5 occurs in the center of the upstream vortex. The ratios in the downstream vortex range between 0.1 to 0.3, which are higher than in the cold flow.

The core of the primary jet has a ratio of about 0.1 as it penetrates into the flowfield. As in the cold flow, a local minimum of near zero is seen in the primary jet flow above the centerline of the combustor. The portion of the primary jet flow that reaches the fuel injector dome has a slightly higher ratio than in the cold flow. However, the primary flow above the secondary recirculation and the vortical flow within the secondary recirculation zone have slightly lower ratio in the hot flow than in the cold flow. Just before the entrance of the primary flow into the secondary zone, the ratio reaches a high of about 0.3 above the centerline. Although the ratio at the top dilution jet inlet is also about 0.6, it decreases much faster than in the cold flow. A local minimum of zero is found where the splitting primary flow crossflows with the strong dilution jet. In the lower part of the dilution jet flow, the ratio is slightly higher in the hot flow. At the exit plane, the ratio varies between 0.1 and 0.3, which is higher than those present in the cold flow.

**Hot Flow Section 2**

The ratio of the shear stress and the turbulence kinetic energy in Section 2 is shown in Fig. 149. In the primary vortex, the ratio is high near the top liner at about 0.4 and decreases to 0.1 in the lower edge of this vortex. Just upstream near the fuel injector, the ratio is low, about 0.1. However, fairly high ratio is within one inlet diameter downstream of the injector inlet probably due to the high shearing action of the fuel jet near the inlet. From the point where the fuel jet enters this plane to slightly above the bottom liner, the ratio increases from 0.1 to 0.2. When the fuel jet splashes onto the bottom liner, the ratio has increased to 0.3. The ratios in the downstream shear layer of the jet and the vortex upstream are smaller than that in Section 1. In the downstream vortex, there is a local zero minimum.

In the primary jet, the ratio that reaches a high of 0.3 is reached upstream of the flow; it then decreases to a local minimum of zero before it increases again to about 0.4 in the vicinity of the injector dome. In the shear layer downstream of the primary jet, the ratio reaches a peak of 0.5 before decreasing rapidly to about 0.1 in the secondary recirculation. The bottom dilution jet has a fairly small and constant ratio of 0.1 in the core. In the crossflow between the primary flow and the bottom dilution jet, in the upper part of the secondary zone, there is a local maximum of 0.4. In the downstream shear layer of the bottom dilution jet, a local maximum of 0.5 is reached.
Hot Flow Section 3

Figure 150 depicts the ratio of the shear stress and the turbulence kinetic energy in Section 3. In the primary vortex, two local maxima of about 0.3 are found one above and one below the vortex center. Further downstream of the vortex center, a higher local maximum of about 0.4 is seen in the region where the primary flow splits. The ratio in the primary jet is decreasing from about 0.3 near the bottom liner to a minimum of zero below the centerline of the combustor before it increases up to a peak of 0.54 in the primary flow upstream of the secondary zone below the thumb-nail louvre. The ratio in the secondary recirculation is fairly low and constant at about 0.1. In the secondary zone, the ratio is about 0.1 in the core of the bottom dilution jet. On the upstream and downstream of the dilution jet, a local maximum of 0.3 and 0.4, respectively, are found in the lower half of the combustor. The majority of the outflow at the exit plane has a ratio of about 0.1.

6.4.9 Turbulence Intensity TI

Cold Flow Section 1 (Fuel Injector Plane)

The turbulence intensity of the cold flow in Section 1 is shown in the contour plot of Fig. 151. In the primary flow around the ignitor, the turbulence intensity is low at about 0.5. The intensity increases from the wall towards the center of the primary vortex to intensities of about 3.0. Towards the upstream shear layer of the fuel jet, the intensity decreases. In the core of the fuel jet, the intensity is seen ~0.75 near the inlet and as the fuel jet penetrates across the primary zone, the intensity remains quite constant and have values ranging from 0.5 to 0.75. As it strikes the bottom liner, the intensity increases to near 2.0. The gradient of the turbulence intensity is higher on the downstream than on the upstream shear layer because of the higher velocity gradient between the fuel jet and the primary jet in the upper half of the combustor. A local maximum of 2.8 is seen in the downstream shear layer of this fuel jet. In the vortex downstream of the fuel jet, the intensity has a local maximum of about 3.0.

The core of the primary jet has a local maximum in the intensity of about 1.0 near the inlet. It then decreases to a local minimum of 0.43 as the primary jet flows upward. In the secondary recirculation, there is a local maximum of 9.7 in the center of this vortical flow. Towards the bottom liner, the intensity decreases to about 0.5. A slight increase in the turbulence intensity is seen underneath the thumb-nail louvre. The top dilution jet enters the combustor with turbulence intensity of about 0.5 and increases as it penetrates into the flowfield. It reaches a local maximum of 9.9 below the centerline at where the jet turns toward the exit plane rapidly. A high gradient of the turbulence is observed upstream of the dilution jet where a local maximum of 4.5 is seen. At the exit plane, the turbulence intensity is fairly constant at 1.0 except a slight increase to 2.0 in the turning flow near the bottom liner.
Figure 152 depicts the contours of the turbulence intensity in Section 1 of the hot flow. In the primary vortex, the intensity is slightly higher than that in the cold flow near the top liner. Higher intensity gradient from the fuel jet shear layer are seen extended outward into the primary vortex. At the fuel jet inlet, the intensity is at about 0.5 and it remains fairly constant at this level in the fuel jet core before it decreases further to a local minimum of 0.33 just below the combustor centerline. Then near the bottom liner, it increases up to 1.0 where it splashes. The core of the fuel jet has slightly lower turbulence intensity with combustion than without combustion. This may be due to the larger increase in the jet momentum relative to the rise in the normal stresses. On the upstream shear layer of this fuel jet, a very high local maximum of 15.8 is seen. This is expected to be high since the primary vortical flow is opposing the expanding fuel jet at this location and the velocity magnitude is approaching zero, whereas the velocity fluctuation is high. On the downstream shear layer of the fuel jet, a local maximum of 3.34 is found.

At the primary jet inlet, the turbulence intensity is about 0.5. The primary jet flow has slightly lower intensities than in the cold flow. A local minimum of 0.32 is reached further downstream of the primary jet flow. In the secondary recirculation, the turbulence intensity gradient is lower than that found in the cold flow and the local maximum at the center of the secondary recirculation is at 7.5, which is lower than that in the cold flow. At the top jet inlet, the intensities is slightly below 0.5. At the upstream shear layer of the dilution jet, a local maximum of 5.0 is seen. At where the top dilution jet turns downstream, the turbulence intensity is a local maximum of 11.3. At the exit plane, the intensities are at about 1.0, which are relatively lower than those in the cold flow.

Hot Flow Section 2

The contour plot of the turbulence intensity in Section 2 is shown in Fig. 153. The turbulence intensity in the primary vortex is similar to that in Section 1 in front of the ignitor but has appreciably smaller values in the lower part of the primary vortex because the fuel jet enters the flowfield at a lower portion of the combustor. In about two fuel injector inlet diameter downstream along the fuel jet flow, a decrease in the intensity level is observed from about 1.0 to below 0.75. Further downstream of the jet flow, the intensity remains at about 0.75 at the bottom where the jet strikes the liner. Comparatively, the intensity in the fuel jet core is slightly higher than that in Section 1. The turbulence intensity gradients on both sides of the fuel jet are lower than those in the fuel injectors plane. The intensities in the primary jet flow from the bottom liner to the top liner above the secondary recirculation zone are quite similar to those seen in Section 1. In the secondary zone, higher turbulence intensities are seen. The local maximum in the secondary recirculation is higher but the location of this maximum has shifted downward because of the expanding primary flow. In the secondary zone, the turbulence intensity is about 0.5 near the inlet. A local maximum of 6.0 is on the upstream of the dilution jet. The interaction between this and the top dilution jet generated a local maximum above the centerline at 15.3. At the exit, the intensity ranges between 1.0 and 2.0, with slightly higher intensity in the wake flow.
Hot Flow Section 3

The turbulence intensity in Section 3 of the hot flow is shown in Fig. 154. In the primary vortex, the turbulence intensity is extremely low compared with those in the previous sections, cold and hot flows alike. Along the front dome, the intensity is less than 0.4. Towards the primary vortex center, the intensity level increases rapidly to a local maximum of about 4.0, which is similar to that in Section 2. The rest of the primary flow is basically having an intensity level below 1.0. A high gradient of the turbulence intensity is found where the primary jet influences the flow. The intensity of the primary jet is quite constant at about 0.4. The intensity within the secondary recirculation varies between 0.6 to 4.0, which is similar to those in the primary vortex. The primary flow that enters the secondary zone has an intensity of 0.4, which is slightly lower than that observed in the two other sections of the hot flow. In the secondary zone, the intensity level in the bottom dilution jet increases from 0.2 at the inlet to about 1.0 at the centerline of the combustor. Then it decreases as this dilution jet is forced towards the exit by the massive primary flow in the upper part of the combustor. In the wake flow of the bottom dilution jet, the turbulence intensity is quite even at 1.0. At the exit plane, the intensity decreases from 0.6 near the top liner to about 1.0 near the bottom liner. These intensities are lower than those in the other sections because of the higher magnitude of the exiting velocities in this section.

6.5 Effects of Combustion on the Flowfield

Although there are many activities and developments being done on multidimensional computer modeling of combustors, their applications to the design of practical combustors are restricted due to the fact that important natures of the flame front generation and/or suppression of turbulence by the combustion process are either ignored or improperly modeled. As Ballal (1988) has pointed out, turbulent dilation and viscous dissipation processes suppress flame turbulence, whereas turbulent advection and shear-generated turbulence processes generate flame turbulence. One of the main objectives of this work was to determine the influence of the heat release by the combustion process on the flow characteristics and on the turbulence properties in a combustor. The experimental data obtained by the laser Doppler velocimetry technique in Section 1 (fuel injector plane) of the cold flow with fuel injection and the hot flow are compared below. Measured mean velocities $\bar{u}$ and $\bar{V}$, and the Reynolds shear stress $\bar{u} \bar{v}$ are used in the comparison. However, the two normal stresses $\bar{u}^2$ and $\bar{v}^2$ are lumped together and compared using the turbulence kinetic energy $k$ quantity.

6.5.1 Velocity $\bar{u}$

Comparison of the mean velocity $\bar{u}$ between the hot and cold flow is shown in Fig 155. Hot flow data are shown in solid lines. Near the front-end of the primary zone at the top next to the ignitor, there is an increase in the strength of the small primary vortex. At the centerline, the counter-rotating recirculation upstream of the fuel jet causes $\bar{u}$ to be higher in the positive
axial direction. The negative velocity at the bottom is an indication of the stronger splashes at the liner wall caused by the higher momentum of the fuel jet with combustion. Further downstream in the center of the primary zone, $\vec{u}$ is slightly lower near the top due to the stronger suction effect of the fuel jet on the surrounding flow. The high momentum of the fuel jet is seen to peak at the same transverse location in the hot flow as in the cold flow. However, the maximum value of $\vec{u}$ for the hot flow is 43% higher than the cold flow. The axial velocity of the flame stabilizing vortex downstream of the fuel jet at the bottom is slightly higher. The axial velocity near the fuel injector is quite similar except that the hot flow has a higher peak value right at the injector inlet. Near the centerline below the injector, the primary jet is also found to have slightly higher axial momentum. At the bottom next to the primary hole, the U velocity is more negative because of the higher suction created by the stronger primary jet. Downstream of the injector, the mainstream is seen leaving the primary zone with higher axial velocity in the upper half of the sector. The recirculation downstream of the primary jet and the vortex in the secondary zone near the bottom liner wall have their axial velocities slightly higher for the combusting case. The axial velocity is positive across the entire exit plane for both cases with their maxima and minima at about the same transverse locations. The maximum axial velocity for both flows occur at about the centerline and is 39% higher for the hot flow.

6.5.2 Velocity $\vec{v}$

Figure 156 shows the comparison of the transverse mean velocity $\vec{v}$. In the front-end of the combustor, the counter-clockwise rotating primary vortex has a more negative $\vec{v}$ component, and the upstream fuel jet recirculation which rotates clockwise has a positive $\vec{v}$ component (note that this recirculation does not exist in the cold flow). The strong fuel jet impinging on the bottom liner wall is indicated by the much higher negative $\vec{v}$. In the middle of the primary zone (grid 12), the entrainment of the surrounding air into the fuel jet is shown by the slightly larger negative $\vec{v}$. As in the axial velocity, the transverse component of the fuel jet in the center of the primary zone is much stronger with combustion. The maximum for the hot flow is located slightly closer to the centerline and it is about 47% higher than in the cold flow. The zero-crossings of $\vec{v}$ for both hot and cold flows occur almost at the same location which suggests that the size of the flame-stabilizing vortex does not change with combustion; however, the vorticity is much higher for the hot flow as shown by the higher $\vec{v}$ near the bottom liner. At the injector inlet, both the transverse velocity and its transverse gradient are higher with combustion. The overlapping of the zero-crossing of $\vec{v}$ for both flows suggests that the edge of the fuel jet near the fuel injector remains the same with combustion. The higher transverse momentum downstream of the primary jet is found in the flowfield underneath the injector. A very interesting observation is found of the flowfield beyond the injector: $\vec{v}$ does not differ appreciably from the cold flow value. The mainstream leaving the primary zone and entering the secondary zone near the top liner has almost lost its entire transverse momentum, indicating that the flow is near horizontal. The top dilution jet is seen entering the sector with slightly higher $\vec{v}$ velocity but turning toward the exit more rapidly, and the outflow toward the exit plane is also near horizontal.
6.5.3 Turbulence Kinetic Energy $k$

The turbulence kinetic energy of the hot and cold flows are shown in Fig. 157. In the front-end of the primary zone, the increase in $k$ due to the strengthening of the primary vortex near the ignitor is not significant. However, turbulence generated by the upstream fuel jet recirculation, the shearing of the outer edge of the fuel jet, and the strong impingement of the fuel jet at the bottom liner is shown clearly in the lower portion of grid 4. In the middle of the primary zone, more turbulence is found in the inner portion of the flow due to higher velocity gradients of the fuel jet due to temperature effect. Near the injector inlet, the turbulence level of both the cold and hot flows are extremely high. The maximum $k$ shown in this figure is located along the centerline of the fuel injector with the turbulence kinetic energy in the combusting flow 23% lower than in the cold flow. This smaller $k$ for the combusting flow is due to the local laminarization of the flow caused mainly by the increase in the kinematic viscosity of the fuel-air mixture due to an increase in temperature. The enhanced recirculation in the shear layer of the vortex downstream of the fuel jet also increases $k$ near the bottom liner wall.

As the primary jet flows downstream and upward toward the top liner, no appreciable increase in $k$ is found. However, near the edge of the primary jet, the slightly stronger incoming primary jet of the hot flow created a higher velocity gradient across the shear layer and thus a slight increase of $k$ at the edge of this jet. The higher inlet velocity of the primary jet has also moved the location of the maximum $k$ further away from the bottom liner. Further downstream of the primary jet (grid 23), the turbulence level is very low across the entire sector from the main flow near the top to the vortex near the bottom for both flows. At the end of the primary zone, $k$ is higher for the hot flow due to the higher velocity gradients generated by the stronger inter-mixing of dilution jets and the main flow near the centerline. Close to the exit plane, $k$ remains higher for the hot flow with the maximum occurring closer to the top liner wall. The difference between the maximum of the two flows is 40%.

6.5.4 Reynolds Shear Stress $\overline{u'v'}$

Figure 158 compares the measured shear stress $\overline{u'v'}$ of the hot and cold flow experiments. In the front-end of the primary zone, the shear stress is near zero in the primary vortex but increases rather sharply when entering the upstream recirculation of the fuel jet region, and reaches a peak at the edge of the fuel jet. Then the shear stress decreases inside the fuel jet, but increases back up again to about the same maximum level at the bottom wall due to the strong impingement. In the middle of the primary zone, the shear stress is higher in the hot flow near the top liner due to stronger entrainment. From the edges of the fuel jet, shear stress increases to a relative maximum within the jet and then decrease to a relative minimum at about the centerline of the fuel jet. This profile of the shear stress is the same for both the hot and cold flows, except that the hot flow is 43% higher at the peak. The relative maxima on either side of the centerline of the fuel jet is not symmetrical because the grid line is cutting across the fuel jet obliquely instead of normally to the centerline of the fuel jet, and also due to the different flow features upstream and downstream of this fuel jet. On the downstream of the flame-stabilizing vortex, the shear stress is again near zero for both flows.
At the injector inlet, hot flow has a much smaller shear stress than the cold flow due to the local laminarization of the fuel jet by the increase in temperature, though both have a peak at the centerline of the fuel jet. The shear generated at the edge of the fuel jet due to stronger entrainment with the surrounding flow for the hot flow is seen further away from the injector. In the center portion below the injector, the upward flow of the primary jet does not create any significant amount of shear stress. The effect of combustion on shear stress immediately downstream of the primary jet is identical to that in the turbulence kinetic energy where the maximum occurs at the shear layer, and the high shear region of the hot flow pushed upward deeper into the flowfield by the higher velocity primary jet. In this recirculation region, the maximum shear stress is higher for the hot flow, but unlike the turbulence kinetic energy, the location of this maximum is nearly the same as in the cold flow. Near the end of the primary zone (grid 23), slightly higher shear stress is found near the top liner, probably convected by the primary jet and the main flow from upstream.

In the recirculation region behind the primary jet, the shear almost vanishes completely. At the end of the primary zone, shear stresses generated by the crossflow of the top dilution jet and the mainstream, and the inter-jet mixing of the dilution jets are higher for the hot flow case. Finally, in the secondary zone, the velocity gradient is higher downstream of the top dilution jet for the hot flow, hence the maximum of the shear stress is 84% higher for the hot flow. The relative maxima and minima of the shear stress in this region near the exit plane occur at almost identical locations in both flows.

6.6 Combustor Flow Structures With Fuel Injection

With the introduction of the fuel injector in the combustor, the dominating primary vortex with low velocities, Reynolds stresses and turbulence as observed in the cold flow without fuel injection is not evident now. In the streakline plot of the fuel injector plane in Fig. 159 for the cold flow with fuel injection, the primary zone is overwhelmed by the strong fuel jet injected from the fuel injector. Because of the quiescent nature of the primary vortex, the high momentum fuel jet finds no hindrance along its path when penetrating across the front-end of the combustor. The flow in front of the ignitor is not yet properly developed into a primary vortex due to the recirculating part of the flow being cut off by the penetrating fuel jet. The splashing of the fuel jet at the bottom liner creates smaller vortices in the front dome. Due to the enhancement of the rotational motion by the fuel jet, the bottom louvre and the primary jet flows, a large vortex is attached to the fuel jet downstream. Part of the primary jet flow is entrained by the fuel jet, part of it enters the wake of the fuel injector, another part recirculates in the secondary recirculation zone; the rest enters the secondary zone via the upper part of the combustor. The secondary recirculation zone is much larger in size than that in the flow without fuel injector since much of the primary flow from this plane spreads to the neighboring planes in the tangential direction. The top dilution jet does not splash as hard onto the bottom liner as in the case without fuel injection. The high rotational flow of the dilution jet at the centerline of the combustor is due to the interaction with the opposing off-set bottom dilution jets.
The ignition and combustion processes inside the combustor can be described in the following manner. Fuel in the center annulus co-swirls with air in the outer annulus of the fuel injector. With the modified Parker-Hannifin fuel injector, additional fuel is premixed with air before being injected through the center core. The swirling jet encourages entrainment of the fuel-air mixture with the surrounding flows in the primary zone. As shown in Fig. 159 of the cold flow, part of the swirling fuel jet near the injector entrains fresh air and is drawn into the upper front of the primary flow where more fuel-air mixing occurs and this mixture flows with a low velocity directly in front of the ignitor. As ignition is turned on, pockets of near stoichiometric mixture in the vicinity of the ignitor are ignited and the flame propagates to regions around the fuel jet where the rich fuel is properly mixed with air and has fuel/air ratio within flammability limits.

The streakline plot of the flowfield in Section 1 with combustion is shown in Fig. 160. Because this combustor is designed to have the majority of the fuel-air mixture in the lower portion of the front dome, the upper portion of the primary zone allows air to be preheated through reaction and entrainment of the recirculating hot gas. Part of the flow upstream of the fuel injector is drawn further upstream into the primary vortex after picking up some fuel from the fuel jet. Due to the stretching action of the vortex, the flow that enters the primary vortex in Section 1 is sucked towards the vortex center, which then transports flow from this section to the neighboring ones. To provide a long time duration for the fuel to preheat, evaporate (not present when gaseous fuel is used) and mix, the fuel jet is oriented to cut across the primary zone rather than being injected horizontally towards the front dome or vertically down towards the bottom liner. The incoming fuel jet entrains hot products that are trapped inside the recirculating primary vortex. Although the primary vortex seems small in Section 1, it gradually develops into a big vortex further away from the fuel jet in the tangential direction.

Inside the fuel jet, the fuel-air ratio is normally too high for combustion and the high transport velocity of the jet has the tendency to quench the flame as well. The entrainment of the surrounding hot gases causes the fuel jet to preheat as well as to decelerate. However, the fuel jet still retains quite an amount of momentum when it impinges onto the liner, and the resulting splashing increase the vortical flows nearby. The liner becomes a means to decelerate the jet and transfer the remaining momentum in other directions.

The louvre air in the primary zone serves three purposes. First, it assists in setting the flowfield and direction of flow. Second, it acts as a cooling film insulating the hot gases from the combustor liners before its mixing with the fuel jet. Third, it is the oxidant which is essential to combustion. As the fuel jet strikes the bottom liner, the splashing action increases the shearing at and above the liner, which in turn speeds up the mixing process and accelerates combustion. By properly metering the amount of air in the primary zone, intense combustion is delayed till the fuel rich mixture reaches the primary jet where excess air is readily available. The high shearing action and the high level of turbulence at the primary jet creates a rapid mix and burn region. The hot combustion product is also lifted away from the liner and directed to the secondary zone where it is quenched by the dilution jets. Also, part of the hot combustion product recirculates back into the primary vortex as seen in the streakline plot of Section 2 in Fig. 161.
Further away from the direct influence of the fuel jet, the primary vortex becomes the dominating feature again in the primary zone as shown by the streakline plot of Section 3 in Fig. 162. However, the recirculation region is shifted much closer to the front dome rather than extended way back in the primary zone as in the cold flow without fuel injection. A large amount of the primary flow is seen leaving the combustor.

Some interesting features are observed from the streakline plots. Although the direction of the vortical flow in the primary vortex is counter-clockwise in all three sections, the vortical flow is being drawn into the center in Section 1 whereas the vortical flow is coming out of the vortex center in both Sections 2 and 3. One may conclude that in the fuel injector plane, flows upstream of the fuel injector are mostly drawn into the primary vortex, and because of the three-dimensional nature of a vortex, the stretching process in fact transports some of the mass flow in this plane to the neighboring planes such as Sections 2 and 3.

Consider now the pair of vortex attached to the fuel jet near the bottom liner. The vortex upstream and downstream of the fuel jet in both the cold and hot flows rotate in the clockwise and counter-clockwise directions, respectively. Contrary to the vortical flow characteristics of the primary vortex, the vortex upstream is consistently drawing flow towards the vortex center in both Sections 1 and 2, and the vortex downstream is also seen consistently having flows coming out of its center in both Sections 1 and 2. These indicate explicitly that the vortex upstream and the vortex downstream seen in these two-dimensional mappings of the flowfield are not two separate vortices, but they are the same vortex that stretches and encircles the fuel jet cone. The direction of stretching in this vortex can be determined by considering the stretching orientation of the primary vortex. Since the primary vortex is stretching in the $+y$ direction according to the right hand rule, the vortical flow upstream is then also stretching in the $+y$ direction whereas the vortical flow downstream is stretching in the $-y$ direction. The rotation of the fuel jet seen in the flow visualization video tape in slow motion agrees with the direction deduced above.

The secondary recirculation zone also has the vortical flow being drawn in towards the vortex center in one plane and out the other. It is believed that the three-dimensional flow nature of the secondary recirculation in the flow with fuel injection is very similar to that in the flow without fuel injection. In the secondary zone of Section 2 (see Fig. 161), the rotational flow at the upstream shear layer of the bottom dilution jet caused by the top dilution jet in a plane near the fuel injector is clearly seen. This flow feature is not evident any more in planes that are further away from the fuel jet because higher mass flows are leaving the combustor and the stronger crossflow suppresses this rotational flow from reoccurring (refer to Section 3).
Chapter 7

Computational Results

From the experimental results discussed above, the complex flowfield inside the combustor is truly three-dimensional. To simulate such flow with a two-dimensional computer program, some assumptions have to be made. The tangential louvre located at the top liner wall near the end of the primary zone is not included in the simulation, and the truly 3-D combustor flow is treated as a 2-D flow in the numerical analysis.

7.1 Cold Flow Without Fuel Injection

7.1.1 Predicted Streamlines

One set of computation was done for each of the three measured sections with the assumption that the flow in a section is established by the inlet velocities measured in that particular plane only, and that flows from other planes have no influence on the local plane. Using this approach, the measured values of $\bar{u}$, $\bar{v}$ and $k$ close to the inlets were extrapolated to the inlet ports, and these values were used as the inlet boundary conditions to simulate the flow in the three separate planes. Hence, three sets of results were obtained, one for each plane. The streamline values have been non-dimensionalized by the maximum streamline values at the exit plane.

Section 1 (Top-Jet Plane)

The computational streamline of the Top-Jet Plane is shown in Fig. 163. The recirculating flow in the primary zone has an elliptical shape due to the deflection of the oncoming vortical flow by the primary jet. Just downstream of the primary jet, there are three secondary recirculations. The small vortex attached to the primary jet and the large vortex are created by the sudden expansion of the jet. These vortical flows are separated by a third vortex, which might be created due to the stairstep boundary of the bottom liner at that location.
The top dilution jet has a deep penetration into the combustor and thus causes another large recirculation in its wake.

The streamlines obtained from the computation bore only some resemblance of the measured flowfield. It correctly predicted the primary recirculation zone of elliptical shape, and the deep penetration of the top dilution jet. However, the extent of the primary vortex is slightly longer than measured, the primary jet does not create three vortices, the top dilution jet should penetrate right across and splash on the bottom liner without much turning towards downstream, and the recirculation in the wake of the top dilution jet is not evident in measurements. The prediction of this section is poor from the primary jet on downstream.

Section 2 (Inter-Jets Plane)

The computed streamline of Section 2 is shown in Fig. 164. There are many flow characteristics that can be identified with the experimental results. A large counter-clockwise rotating primary vortex is predicted in the primary zone. However, the center region is predicted to be much closer to the front-end than measured, and its shape is more circular. This may be due to the fact that the influence of the primary jet from the Top-Jet Plane is much more stronger, and the mass addition of the primary jet from the neighboring planes into this one inside the flowfield is not predicted by the present 2-D code. The secondary recirculation is lying low above the bottom double-band louvres. Due to the massive outflow from the primary zone into the secondary zone, the secondary recirculation is thus elongated by it in the longitudinal direction. In the dilution zone, both the top and bottom dilution jets penetrate a short distance into the main flow and turn downstream. The exiting flow from upstream is thereby squeezed in between in order to leave the combustor. A recirculation region is seen in the wake of the bottom dilution jet and a much smaller one is seen downstream of the top dilution jet because the top wall sloped in and reduced the void area of the wake. The LDV measurements in the dilution zone looked quite different from the predicted results. The spreading of the dilution jets were very difficult to simulate. The neglect of these dilution jets would result even in more unrealistic results. The measured jet velocities near the jet inlets were used in the numerical simulation as the boundary conditions for the jets. These inlet jet velocities were given components normal to the liners and they exhibited a lack of penetration, especially the top dilution jet. The characteristics of the flow in the dilution zone are predicted results from a 2-D code since all inflow must leave via the same plane.

The predicted flowfield of Section 2 is by far more realistic than the prediction made for Section 1. Many of the flow characteristics were observed in the experimental results, except for the extra recirculations downstream of the dilution jets in the secondary zone. More detailed comparisons will be shown later.

Section 3 (Bottom-Jet Plane)

The computed streamlines of the Bottom-Jet Plane is shown in Fig. 165. The primary recirculation region is predicted very similar to that of the Inter-Jets Plane, which is quite
circular in the front-end and more elliptical near the end of the primary zone. Both this and the last sections have less elliptical primary vortex because the primary jet flow added onto these planes are not appropriately accounted for in the 2-D code. Downstream of the primary jet is a secondary recirculation whose shape is well predicted. A very small vortex is also found in the shear region between the secondary recirculation and the bottom dilution jet. The primary flow is first lifted upward by the primary jet in the primary zone, and then pushed further upward towards the top liner by the bottom dilution jet in the secondary zone. A small vortex is found in the wake of the bottom dilution jet. At the exit, a much larger recirculation is predicted due to the high velocity of the bottom dilution jet, which created a low pressure region downstream of it.

The predicted Bottom-Jet Plane has most of the flow characteristics measured in Section 3. The primary vortex is less elliptical and its center is predicted too low and too close to the front-end of the combustor. The impingement of the bottom dilution jet onto the top liner is not well predicted, and the large recirculation created by the bottom dilution jet is not evident in the LDV measurements.

**Hypothetical Plane**

Another approach which was adopted to predict the flowfield was to simulate the flow by superposing the three planes in one. That is, all the inlets were assumed to exist at the same plane. The extrapolated inlet flow properties from measurements of Sections 1 and 2 were used as the inlet conditions for this Hypothetical Plane.

The predicted streamlines of this plane are shown in Fig. 166. The recirculating flow in the primary zone is very well simulated. It showed that a large oval shaped toroidal flow reversal is located in the primary zone with the downstream side being compressed toward the top liner wall by the primary jet. The center of the primary vortex measured from the front end of the combustor in the axial and transverse directions are (36.83 mm, 11.43 mm) for Section 1, (35.56 mm, 15.24 mm) for Section 2, (33.02 mm, 15.24 mm) for Section 3. Computed results of the Hypothetical Plane give (38.1 mm, 13.21 mm) as the location of the center of the primary vortex which is in excellent agreement with the experiments. Had the set of tangential louvres been modeled, more mass would enter the combustor, forcing the downstream end of the primary recirculation zone further upstream, and the center of the recirculation might be even better predicted.

The trajectory of the primary jet and the secondary recirculation zone immediately downstream of it are also very well predicted. In the secondary zone, the actual flow is truly 3-D and the present 2-D code was unable to simulate the crossflow of the mainstream and the dilution jets, and the interaction of the top and bottom dilution jets which oppose each other directly. Since all mass that enters the flow domain must exit in the same plane in a 2-D code, the primary flow is expected to leave the secondary zone about the centerline. Above the primary jet flow is the top dilution jet entering the flow domain with a lack of penetration into the main flow. Below the primary jet flow is the bottom dilution jet with slightly more depth of penetration into the flow domain than the top dilution jet. Moreover, the two opposing dilution jets simply turn towards the exit, resulting in no inter-jet mixing nor crossflow. The
small recirculation downstream of the top dilution jet is not present in the measurements and the recirculation downstream of the bottom dilution jet is also overpredicted. It is expected that all the turbulence and shear stresses generated in the secondary zone would not be well predicted.

\[ \text{7.1.2 Comparisons of Flow Properties} \]

The well predicted streamlines in the primary zone of the Hypothetical Plane indicated that the influence of the primary jet was substantial. However, the computed results of the Hypothetical Plane would obviously not be appropriate to use for comparison with any of the measured planes, but to illustrate the point indicated above. Since the streamlines were not quite well predicted for Sections 1 and 3, the measured flow properties were only compared for Section 2 with those predicted in the Inter-Jets Plane. The measured and predicted longitudinal velocity \( \bar{u} \), traverse velocity \( \bar{v} \), turbulence kinetic energy \( k \), and shear stress \( \bar{u}'\bar{v}' \) are compared below. In the comparison plots, the experimental measurements are shown with lines connected to the zero reference and the numerical predictions are drawn in bold solid lines. At the bottom of the figure are the locations of the grids used in the experiments. The scale of the plotted data is also given in the figure.

\[ \text{Velocity } \bar{u} \]

Fig. 167 compares the measured with the predicted longitudinal velocity \( \bar{u} \). The characteristics of the toroidal flow reversal in the primary zone are clearly indicated in this figure. The vortex rotates in a counter-clockwise direction. The primary vortex reaches as far downstream as the top tangential louvre as shown by the upper portion of grid line 23. The measured \( \bar{u} \) in the primary zone has the characteristics of a thick layer of high velocity flow close to the liner wall and a rather linear velocity profile in the core of the recirculation. It is this linearity that makes the recirculation seem like a solid body rotation. The recirculation here is set up by the combustor dome in the front-end, and the locations and directions of the wall louvres. The numerical model predicted a thinner layer of high velocity wall flow and lower velocities than the experiment and a non-linear but near zero velocity profile in the core of the recirculation. This may probably be due to the fact that the actual velocities are higher at the louvre inlets than the values used for boundary conditions in the code. It may also be caused by (numerical) false diffusion. The predicted zero-crossings of \( \bar{u} \) in the primary vortex agrees very well with the measured results. Near the inlet of the primary jet, velocity \( \bar{u} \) is underpredicted. This can be easily explained by the fact that the boundary condition for the primary jet is given only the traverse component. Since the primary jet originated in Section 1, it must have attained a longitudinal velocity component when it expands and reaches Section 2. Good agreement is found from the front-end up to the rear-end of the primary zone, about the top and bottom double-band louvres. The locations and magnitudes of the maximum and minima near the end of the primary zone matched almost exactly. Further downstream in the secondary zone where the influence of the dilution jets was inevitable, the agreement deteriorates. The penetration of both the top and bottom dilution jets are indicated by the
drop in magnitude at the centerline of the LDV measurement. The escaping of the main flow and the spreading of the jets near the top and bottom liner are also seen in the measured data. However, in the numerical results, the top maximum corresponded to the turning of the top dilution jet just after its entrance into the combustor. The centerline maximum is the consequence of the compressing action of both dilution jets on both sides of the main flow, thus speeding the flow in the centerline. The bottom maximum corresponds to the turning of the bottom dilution jet. The interfaces of the dilution jets and the mainstream are shown by the two relative minima on the curve. The realism of a crossflow and the intermixing of jets can only be simulated with a 3-D code; better agreement can then be expected.

**Velocity \( \vec{v} \)**

The measured and the predicted transverse velocity \( \vec{v} \) are compared in Fig. 168. The first two plotted data lines in the primary zone indicate that the velocity \( \vec{v} \) is flowing downward with higher velocity near the front-end liners. The agreement in the front-end is excellent. The predicted zero-crossing of \( \vec{v} \) in the primary vortex deviates slightly from the measured one, as seen in grid 7. Further downstream, near horizontal flow is predicted near the top liner resulting in near zero \( \vec{v} \). Towards the lower part of the combustor around the primary jet, velocity \( \vec{v} \) is overpredicted. As discussed for the case of the velocity \( \vec{u} \) above, the boundary condition of the primary jet was only given a transverse velocity component, hence higher predicted values of \( \vec{v} \) are observed both upstream and downstream of the primary jet. It is speculated that if the primary jet inlet trajectory were oriented more downstream, better predictions would be obtained for both \( \vec{u} \) and \( \vec{v} \) velocity components. Very good agreement is seen at the end of the primary zone about the top and bottom double-band louvers. The prediction of the primary jet flow above the secondary recirculation is excellent. Near the bottom liner, the predicted secondary recirculation zone is slightly larger and more elliptical than the measured one. Hence, a negative velocity in \( \vec{v} \) is obtained. In the dilution zone, the predictions agree poorly with the measured values. Near the top liner, in the top dilution jet, very high negative velocity \( \vec{v} \) near the inlet is predicted; whereas in the real flow the top dilution jet penetrates into this plane further away from the jet inlet as it cuts across the flowfield. Note that the predicted maximum value is only 4 m/s higher than the measured peak. One has to realize that, in the present 2-D code, it is impossible to insert a boundary condition which can simulate the addition of mass flow in the inner region of the flow domain as in the case of the actual cross-plane flows. Below the centerline, the bottom dilution jet is also poorly predicted. The present code was unable to solve the interaction of the top and bottom jet flow, as seen in the experiment.

**Turbulence Kinetic Energy \( k \)**

The comparison of the measured and predicted turbulence kinetic energy \( k \) is shown in Fig. 169. In the primary zone, the measured \( k \) is slightly higher near the top liner than near the bottom liner due to the center of the primary vortex being located well above the centerline closer to the top liner and compressing the top flow. Hence, higher velocity gradient
and consequently higher turbulence exist near the top liner. The predicted values are much lower in most parts of the primary zone. The general trend of the prediction is that $k$ is high near the liner and decreases to slightly above zero in the inner flow. The measured amount of turbulence immediately downstream of the deflection jet is much higher than the predicted one, although both had similar behavior in that a sharp jump is encountered near the exit of the jet. Near the end of the primary zone, the predicted $k$ is also lower than that measured near the top liner, but the agreement improves significantly in the recirculation region downstream of the primary jet. No comparison is shown in the secondary zone due to the fact that inter-mixing and crossflow processes of the jets are poorly predicted. From the predicted results, the inlet turbulence does not seem to influence the inner flowfield as much as indicated by the sharp peaks near the boundaries. Therefore, the turbulence level within the flow domain is not strongly governed by the boundary conditions but by the source terms in the partial differential equations of $k$ and $\epsilon$. Some explanations of the discrepancies can be given. The 2-D code can only predict flows based on the conditions given on a specific plane. Flows leaving and entering the plane are not accounted for, and all the mass that enters must exit in the same plane. Therefore, flows through gaps between jets, jet mixings, and crossflows are not correctly predicted. Consequently, the shear stresses and turbulence generated in the flow were not properly predicted. Moreover, the standard $k$-$\epsilon$ model does not account for the effect of streamline curvature and turbulence due to pressure gradient in the recirculation regions. The coefficients employed in the model are not universal for all types of flows. In the complex flowfield of a combustor, several of the coefficients should be functions of the flow instead.

**Shear Stress $\overline{u'v'}$**

The comparison of the measured and predicted shear stress $\overline{u'v'}$ is shown in Fig. 170. Although the numerical code with the $k$-$\epsilon$ model does not give $\overline{u'v'}$ as a direct input, the shear stress can be calculated from the predicted $\bar{u}$ and $\bar{v}$ velocity fields using the Boussinesq approximation. The shear stress in the primary zone is not very large in value. Typically, higher $\overline{u'v'}$ are measured in the upper part of the primary zone. The predicted values are all lower than the measured values. They are also mostly near zero in the primary zone, except near the liner. At the end of the primary zone, between the top and bottom double-band louvers, the agreement between the predicted and measured is poor, even though the agreement here for $\bar{u}$ and $\bar{v}$ are excellent as shown earlier. In the secondary recirculation zone, the eddy viscosity hypothesis does not seem to have the same sign as the experimental results, and the predictions here are poor. No data comparison is shown in the dilution zone because of the incorrect prediction of the dilution jets’ trajectories, and the errors in this zone are very high.

### 7.2 Cold Flow With Fuel Injection

The numerical prediction of the combustor flowfield with fuel injection was performed for Section 1 (Fuel-Injector Plane) with the assumption that the flow in that plane is established.
by the inlet velocities measured in that plane only and with no influence from the neighboring planes.

The predicted streamlines in Section 1 are depicted in Fig. 171. The streamline values are non-dimensionalized by the maximum value at the exit plane. In the primary zone, there is a small vortex near the top liner upstream of the fuel injector with a counter-clockwise recirculation. Below it there is a large vortical flow which has a clockwise rotation. Hence, the front-end and bottom louvre flows are predicted to turn 180° rapidly as they enter the flowfield. They are eventually entrained downstream by the strong fuel jet. Upstream of the primary jet, a smaller vortex rotated in a counter-clockwise direction. The fuel jet from the fuel injector entered with an orientation of about -45° towards the bottom dome. However, the fuel jet turns rapidly downstream with very little penetration towards the front dome. It flows towards the secondary zone in the lower part of the combustor. A large low pressure region is created downstream of the fuel jet and the recirculation there is in the counter-clockwise direction. The primary jet is pressed low and flows along the bottom liners all the way to the exit, and no recirculation is seen immediately downstream of the primary jet. The top dilution jet penetrates deep into the combustor to about the centerline before it is forced to turn by the massive outflow near the bottom. A large recirculation is formed in the wake of the top dilution jet which is flowing in a counter-clockwise direction.

These predicted flow characteristics are very different from those seen in the LDV measurements of Fig. 75. The failure to predict the actual flowfield in the fuel injection case is due to the treatment of a truly 3-D flowfield by a 2-D computer code. Since all the mass that enters the flow domain must exit via the same plane with a 2-D analysis, the fuel jet can only drive a large vortical void in the primary zone, and the fuel jet itself has to find a path leaving the flow domain. It is not surprising that the fuel injection case is predicted poorly.

The total velocity \( V_t \) of the predicted flow is shown in Fig. 172. Most of the flow velocities in the primary zone are below 10 m/s. Relatively lower velocities are found in the large vortex in the primary zone. The fuel jet has an inlet velocity of about 107 m/s and high velocity gradients are found along its shear layers. The primary jet has an inlet velocity of 60 m/s and part of its momentum is transported upstream by the small vortex upstream of it. As the fuel jet and the primary jet merged and co-flowed towards the dilution zone, a local maximum of about 75 m/s results. Between the fuel jet and the top dilution jet, another void region is observed with velocities below 15 m/s. The top dilution jet enters at 65 m/s with high velocity gradient along its shear layers. Another low velocity region is found in the wake of the dilution jet.

The mixture fraction \( f \) is equal to the fuel mass fraction \( m_{fu} \) under no combustion condition. The fuel mass fraction in Section 1 is shown in Fig. 173. Since the fuel injector is a pure air-blast nozzle, the fuel is sheared by the center-core air as well as by the outer-annular air. the fuel mass fraction \( m_{fu} \) applied at the boundary of the fuel injector location is 0.4. It is seen that very little amount of fuel is in fact transported to the primary zone. The fuel mass fraction \( m_{fu} \) in the primary zone is below 0.06. A much higher gradient of \( m_{fu} \) is found in the upstream shear layer of the fuel jet than on the downstream one. However, \( m_{fu} \) decreases down to near zero at about the primary jet near the bottom liner. Upstream of the top dilution
jet, a high gradient of $m_{fu}$ exists since $m_{fu}$ is zero in the dilution jet at its inlet.

If this distribution of $m_{fu}$ were true, then the most intense combustion would be at the interface of the fuel jet and the primary jet, as well as on the shear layer upstream of the top dilution jet; this might result in high temperature or even burn through the liner at the top and bottom combustor walls. Also, the dilution air predicted here is used for combustion rather than for its intended purpose of cooling the hot exhaust products. The temperature profile at the exit plane would be absolutely unacceptable for the turbine blades.

### 7.3 Three-Dimensional Computer Simulations

From the streamline and contour plots of the total velocity and fuel mass fraction in the fuel injection case, one would have no reservation to point out that a numerical simulation of a 3-D flow cannot be accomplished by using a 2-D analysis. Due to this severe limitation, no further effort was pursued in simulating the LDV measured sections for the combusting case.

Numerical simulation of the cold flow without fuel injection was performed on the present PW209T combustor by Lai (1990) using a 3-D TEACH type computer code with $k-\varepsilon$ turbulence model. Detailed comparisons of the predictions with the present LDV measurements was also given. A brief discussion of the predicted velocity vector plots is given below.

**Section 1 (Top-Jet Plane)**

The LDV measured and the predicted velocity vectors in Section 1 are shown in Fig. 174. The primary vortex is very well predicted, although the center is slightly more upstream than the measured one. The velocity magnitudes in the primary vortex are slightly higher than measured. The flow trajectory of the primary jet and the characteristics of the secondary recirculation zone are very similar to the LDV data. The top dilution jet is penetrating slightly more upstream and influencing the primary jet flow as well as the secondary recirculation. The expansion of the jet on the downstream near the top liner is not very well predicted, probably due to the fact that the velocity profile was given a constant value across the entire inlet area. Higher velocities are also predicted in the lower part of the combustor in the secondary zone. The low velocity region due to the opposing dilution jets' interactions beneath the centerline is not quite the same as measured. More flow leaving along the bottom liner is predicted.

**Section 2 (Inter-Jets Plane)**

The velocity vector plot of Section 2 is shown in Fig. 175. The primary vortex in this section looked very similar to the last section. The vortex central region is more circular instead of elliptical as seen in the LDV measurement. The velocity magnitudes are also in general higher with the predicted results, most likely because the higher inlet values are used. The trajectory of the primary flow above the secondary recirculation is steeper than measured, and the shape of the secondary recirculation zone is not much different from the one predicted.
in Section 1. However, the LDV data has a more elliptical secondary recirculation zone. In the dilution zone, the spread of the top dilution jet is seen with its orientation more vertical and smaller in magnitude than measured. The spread of the bottom dilution jet has deeper penetration and is more vertical than the measurements. Hence, there is a lower velocity region in the wake of the bottom dilution jet.

Section 3 (Bottom-Jet Plane)

The velocity vector plot of Section 3 is shown in Fig. 176. The primary vortex is well predicted, but the primary jet lacks the momentum as measured by LDV. The secondary recirculation is still maintaining an oval shape as predicted in the previous two sections, whereas the measurement has an elliptical zone that lies along the bottom liner. The predicted velocity magnitude in the primary flow entering the secondary zone near the top liner is lower than the measured one, again probably due to poor prediction of the primary jet flow. Upstream of the bottom dilution jet, more of the primary flow is drawn into the bottom dilution jet than the measurement indicated. The trajectory of the bottom dilution jet is very well simulated. In its wake, some of the flow from the neighboring planes is seen entrained back into the high momentum bottom dilution jet. A small recirculation in the wake of the bottom dilution jet about the centerline is predicted, but is not found in the LDV data.

Overall, the 3-D simulation of the cold flow without fuel injection inside this complex combustor geometry using a TEACH-type computer code with the standard $k$-$\varepsilon$ turbulence model provided very good agreement and realistic solutions of the velocity field. The correct orientations and depths of penetration of the jets, the shapes and lengths of the primary vortex and recirculations, and various flow properties can be well predicted with proper treatment of the boundary conditions at the inlets and fine tuning of the empirical constants employed in the $k$-$\varepsilon$ turbulence model. The excellent agreement of the predicted results with the LDV data points out that the $k$-$\varepsilon$ turbulence model is appropriate in predicting the complex flowfield in a combustor for the velocity field. The present 2-D computer code is very similar in nature to the 3-D code used by Lai. Therefore, the 3-D results substantiated the earlier statement that the poor predictions were due to the incapability of the 2-D code to handle a non-axisymmetric 3-D problem.

However, the 3-D simulation by Lai underpredicted the shear stress $u'\nu'$ and it is most probably due to the isotropic turbulence assumption employed with the $k$-$\varepsilon$ turbulence model, where the LDV measurements indicated that turbulence in the complex combustor flowfield is anisotropic in nature. Less discrepancy between the measurement and the prediction in the dilution zone is observed because the $k$-$\varepsilon$ turbulence model can predict the production of the turbulence kinetic energy quite well, since the interactions of the shear stress and shear strain is dominant in this zone. As Heitor and Whitelaw (1986) pointed out, the mean pressure gradient has a strong influence on the production of the turbulence kinetic energy in the primary zone of the can-type combustor they investigated. Because mean pressure gradient is not taken into account in the production term of the present $k$-$\varepsilon$ turbulence model, the predicted results would underestimated the turbulence kinetic energy and the shear stress quantities.
Chapter 8

Conclusions

The complex flowfield of a toroidal-vortex annular reverse-flow sector-combustor has been studied experimentally and computationally. The experiments employed a two-component laser Doppler velocimetry system which was operated in the dual-beam forward scatter mode to measure simultaneously the longitudinal and transverse components of the mean velocity, and their corresponding turbulence intensities. The experiments were conducted under atmospheric conditions with a constant pressure difference of 204 mm (8 inches) of water across the combustor. By systematic studies, the flow structure within the combustor can be determined.

The first phase of the study was to investigate the flowfield under cold flow without fuel injection conditions. Three sections near the center portion of the combustor were mapped, where the first section cut the top dilution jet, the third section cut the bottom dilution jet nearest to the first section, and the second section was in between the two former ones. The results shown in the velocity vector plots clearly identified the flowfield characteristics—counter-clockwise rotating primary vortex, primary jet with a secondary recirculation attached, the top and bottom dilution jet with deep penetrations and splashings onto the opposing liners, and wake flow downstream of the bottom dilution jet near the exit plane.

The primary vortex is similar in all three sections. Higher velocity magnitudes were measured along the liners due to the fact that flows from the louvres were tangent to the liners. Towards the center of the vortex, the velocity magnitude decreases quite linearly to near zero. Relative to the other flow regions inside the combustor, the primary vortex has very small velocity magnitudes, normal stresses, shear stress and turbulence kinetic energy. The intensities of shear stress and turbulence kinetic energy, however, were rather high in the primary vortex due to the minute velocity magnitude near the vortex center. The normal stresses were also found to be far from isotropic in the primary vortex. Starting from the primary jet, the flow picture depends on the proximity of the primary and dilution jets in that section.

Significantly higher velocities were measured near the inlets of the primary and dilution jets than near the louvres, since the dynamic energy of the outer liner flow was admitted directly into the flowfield; the splash louvres extract dynamic energy as the incoming flow impinges onto the louvre wall before admitted into the flowfield. These jets are essentially free
jets in a crossflow with the main flow. Because of their higher momentum than the oncoming flow, very high velocity gradients and consequently high normal stresses, shear stress and turbulence kinetic energy were identified in the shear layers.

On one hand, the high turbulence that was generated by the jets encouraged rapid mixing of the jet flow with the main flow. Also, due to the suction effect of these high momentum free jets, the surrounding flow was drawn into the jet and entrained. In the dilution zone, this strong entrainment of the wall flow by the dilution jet needs to be cautioned. The purpose of the louvre flows in the secondary zone is for cooling the liners. If the cooling air is stripped off by the dilution jet, the liners will be exposed to high temperature gas products. Therefore, replenishment of the cooling air is essential in the wake region of the jets. The measurements illustrated that there is no recirculation in the wake of the top dilution jet due to the declination of the liner which discouraged any flow separation. However, a wake flow is clearly seen downstream of the bottom dilution jet and attention to the cooling requirement here is strongly advised.

On the other hand, the high momentum jet resulted in deep penetration but poor in mixing, which can cause hot products upstream to reach the exit plane without much dilution and subsequently causing hot spots. Fortunately, much stronger jets that penetrate right across the main flow and create splashing on the opposite liners can effectively increase the mixing rate. The shear stress and turbulence intensities in the core of the jet were low, but it increased rapidly in the shear layers. The results have shown that strong anisotropy is also found around the jet flow regions.

The flow visualization pictures using glass beads and a laser light sheet substantiated the LDV mapped flowfield data.

The second phase of the studies was to investigate the combustor flowfield with the introduction of the fuel jet from the fuel injector under cold and hot flow conditions. The center section of the combustor which cut across the fuel injector core was investigated in both cold and hot flows so that the difference in their flow properties would reveal the effects of heat addition on the flow. Two more sections were also examined for the hot flow in order to obtain a better understanding of the development of the fuel jet in the primary zone. The cold flow results indicated that the prediction of the flowfield without taking fuel injection into account would be very misleading. The understanding of the flowfield characteristics is facilitated by referring back to the first phase of the studies of the cold flow without fuel injection because the primary vortex in the cold flow without fuel injection was a quiescent region with very low velocities and turbulence quantities. With the introduction of the fuel jet, the high momentum jet faced almost no resistance in its flight from the fuel injector face across the primary zone to the front-end bottom liner. The impingement of the fuel jet on the liner created splashing and vortical flows attached to the fuel jet at the bottom liners. The entrainment of the surrounding air into the fuel jet was considered secondary when compared to the rapid mixing due to the splashing at the liners. The fuel jet has drawn much primary flow back into the front end and thus has enabled the primary jet to flow more vertically upwards but without blowing directly across the fuel injector face. The attached secondary recirculation zone was subsequently enlarged.
The louvred air in the primary zone has several functions. The fresh air is guided into the combustor tangential to the liner walls so as to set up the primary vortex direction and sustain the rotational motion of the vortex. It is also used for film cooling along the liners before it is eventually mixed with the fuel and consumed by combustion. Because of the increasing concerns over combustion emissions and their effects on the global environment, stringent regulations on NOx and unburnt hydrocarbons UHC are imposed on future aircraft engines. One approach to meet these regulations in reducing pollutants is by fast burn quick quench combustion processes. The formation of NOx is strongly dependent on the temperature of the combustion product and the residence time under that high temperature condition. By reducing the temperature of the combustion product from the flame temperature of about 2300 K to below 1700 K within a few milliseconds, the level of NOx can be reduced by almost two orders of magnitude. The PW200 series combustor attempts to achieve fast burn quick quench by properly scheduling the amount of airflow to the primary zone so that stoichiometry is not reached in the front end until the flow encounters the primary jet. The rich fuel-air charge mixes rapidly with this jet since it provides large amounts of turbulence kinetic energy for small scale mixing. Because the stoichiometric condition is reached almost instantaneously with the abundance of oxygen from the primary jet, fast combustion is therefore made possible. Since the secondary zone is immediately downstream, the hot combustion product is quickly quenched by the two sets of opposing dilution jets.

The influence of the fuel jet in the primary zone affected the behavior of the dilution jets in the secondary zone. The deep penetration of the top dilution jet and its splashings at the bottom liner was not quite evident directly downstream of the fuel jet. The two sets of opposing dilution jets created a rotational flow at their interface and the top dilution jet was found changing its downward trajectory rapidly towards downstream about the centerline of the combustor.

With combustion, the flowfield characteristics did not differ drastically from that of the cold flow with fuel injection. The most obvious changes were that the vortices throughout the entire flow domain were much more enhanced and more well defined due to the increase in vorticity. More significant Reynolds stresses were found with the fuel jet than with the primary or the dilution jets. Because of the high stress gradients in the jets, very high turbulence kinetic energy was observed, especially with the fuel jet. Highly anisotropic flow regions were also evident in the flowfield. Hence, the assumption of isotropic normal stresses in the $k-\varepsilon$ turbulence model is not well justified.

With a short combustor, it is advantageous to have fuel injection from the radial location. The fuel must then take a path traveling towards the front end and it will be decelerated by the wall as it impinges onto the liner. It is thus reaccelerated in the primary zone by the vortical flows towards the secondary zone. The fuel, in fact, makes two passes through the front end of the combustor and thus the effective length of the primary zone is doubled. This double pass of the fuel flow in the primary zone assists in the preheating and evaporation of the incoming fuel jet. Since the combustor in reality runs with liquid fuel, the vortices in the primary zone provided the mechanism of trapping excessively large fuel droplets which may otherwise form hot streaks inside the combustor and create hot spots or even melt down the combustor liners, and it may pass through the combustor with a very high temperature profile and gradient at
the exit plane which is totally unacceptable to the engine components downstream.

The measured data of the two components $\bar{u}$ and $\bar{v}$ of the mean velocity, turbulence kinetic energy $k$ and the shear stress $\overline{u'v'}$ of Section 1 under cold and hot flow with fuel injection were compared to determine the effects of heat addition by combustion on the turbulence and flow properties of the flowfield. The comparisons showed that heat addition by combustion intensified all the vortical flows and recirculation regions due to a decrease in the density of the gas mixture, locally laminarized the fuel jet near the fuel injector inlet by the increase in kinematic viscosity of the fuel-air mixture, increased the momentum of the fuel jet and its entrainment with the surrounding flow, increased the magnitudes of the turbulence kinetic energy and the Reynolds stresses at the shear layers, and increased the longitudinal velocity, turbulence kinetic energy and stresses of the main flow while decreasing the magnitude of the transverse velocity in the secondary zone.

The numerical modeling has successfully demonstrated that the TEACH-type computer program does provide physically realistic prediction of the turbulent recirculating flowfield of the combustor under cold flow without fuel injection. The flow characteristics and flow properties in the primary zone were quite adequately predicted. However, the present 2-D computer code cannot predict the inter-mixing, crossflow, depth of penetration and splashing of the jets on the liners realistically since the actual flow characteristics are 3-D in nature. To properly resolve all the flow features and obtain realistic flow properties of the complex flowfield, a 3-D computer code is absolutely crucial. This has been successfully demonstrated by some computational fluid dynamicists in the field of combustor flow modeling. Because the three dimensionality of the flow is the dominant factor in the realism of the flowfield prediction, the performance of the standard $k$-$\varepsilon$ turbulence model becomes difficult to be evaluated. However, from the results of the properly predicted primary zone of the cold flow without fuel injection, the standard $k$-$\varepsilon$ turbulence model predicted the $\bar{u}$ and $\bar{v}$ that agreed well with the measurements. However, $k$ and $\overline{u'v'}$ are underpredicted most likely due to the assumption made in the $k$-$\varepsilon$ model that turbulence is isotropic. Similarly conclusion is made by Lai (1990) who simulated the same combustor flow using a 3-D computer code with $k$-$\varepsilon$ model.

The standard $k$-$\varepsilon$ turbulence model is the most widely used model for internal flows that provides a feasible method of predicting engineering flows. But its performance gets poorer with attached, recirculating, swirling, and reactive flows, in that order. The poor performance of this model may be attributed to the neglect of the effects of the streamline curvature, the error of the hybrid scheme in which one arbitrarily neglects the diffusion effects for cell Peclet numbers exceeding 2. The $k$-$\varepsilon$ model also assumes that the Boussinesq hypothesis holds in all types of flow. But it has already been shown that there are small regions of countergradient transport, that negative turbulence kinetic energy production exists in flows such as wall jets, perturbed shear layers, wake flows and asymmetric channel flows where the momentum transport consists of gradient diffusion by small scale eddies and convection by large scale eddies. However, Nikjooy and So (1987) argued that because the algebraic stress and Reynolds stress models do not give satisfactory predictions of the Reynolds stresses, the $k$-$\varepsilon$ model is just as competitive in turbulent flow predictions. In view of the additional computational time associated with higher order closures, the effectiveness of turbulence model prediction
could be obscured by factors such as inlet and boundary conditions, oscillatory phenomena and numerical diffusion, and they recommended that the $k-\varepsilon$ model should remain as the model of choice for calculating flows in practical combustors.

The present study gathered many detailed mappings of the flow properties and turbulence quantities in a practical combustor which serves as a database for the understanding of the flow characteristics and structures of the complex combustor flowfield. Also, this database can be used for the evaluation of the validity of numerical models, and flow models such as turbulence models and combustion models. These in turn will inevitably benefit the future design, research and development of gas turbine engines and analytical modeling of complex flows.

Clearly, the investigation reported here represents the current trends and ongoing efforts in practical combustor flow diagnosis. There are many other aspects that will remain to be studied in detail. A more extensive and careful setup of the LDV system or the employment of a laser two-focus L2F system can be used to investigate the boundary layer flow inside the combustor which will provide valuable information for proper specifications of boundary and inlet conditions in numerical modeling. The number of fuel injectors can be increased for the study of the flowfield characteristics in between fuel jets with combustion. Laser-induced fluorescence LIF and the laser scattering techniques of Mie, Rayleigh and Raman can be used for 2-D planar measurements of temperature, density, species concentration and flow visualization. To study the fuel spray characteristics and the flowfield aerodynamics with liquid fuel, phase Doppler particle anemometry PDPA technique can be employed.
References


FLUENT is a trademark of Creare Research and Development, Inc.


Canada.


PREACH is a trademark of United Technologies Corporation, Pratt and Whitney Engineering Division.


Appendix

Governing Equations

The governing transport partial differential equations for two-dimensional turbulent flows in Cartesian coordinates are:

\( \dot{u} - \) momentum

\[
\frac{\partial}{\partial t} (\bar{p} \dot{u}) + \frac{\partial}{\partial x} (\bar{p} \dot{u} \dot{u}) + \frac{\partial}{\partial y} (\bar{p} \dot{v} \dot{u}) - \frac{\partial}{\partial x} \left( \mu_{eff} \frac{\partial \dot{u}}{\partial x} \right) - \frac{\partial}{\partial y} \left( \mu_{eff} \frac{\partial \dot{u}}{\partial y} \right) =
\]

\[
- \frac{\partial \bar{p}}{\partial x} + \frac{\partial}{\partial x} \left( \mu_{eff} \frac{\partial \dot{u}}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_{eff} \frac{\partial \dot{v}}{\partial x} \right) - \frac{\partial}{\partial x} \left[ \frac{2}{3} (\bar{p}k + \mu_{eff} \text{div} \vec{V}) \right]
\]

\( \dot{v} - \) momentum

\[
\frac{\partial}{\partial t} (\bar{p} \dot{v}) + \frac{\partial}{\partial x} (\bar{p} \dot{u} \dot{v}) + \frac{\partial}{\partial y} (\bar{p} \dot{v} \dot{v}) - \frac{\partial}{\partial x} \left( \mu_{eff} \frac{\partial \dot{v}}{\partial x} \right) - \frac{\partial}{\partial y} \left( \mu_{eff} \frac{\partial \dot{v}}{\partial y} \right) =
\]

\[
- \frac{\partial \bar{p}}{\partial y} + \frac{\partial}{\partial x} \left( \mu_{eff} \frac{\partial \dot{v}}{\partial y} \right) + \frac{\partial}{\partial y} \left( \mu_{eff} \frac{\partial \dot{v}}{\partial y} \right) - \frac{\partial}{\partial y} \left[ \frac{2}{3} (\bar{p}k + \mu_{eff} \text{div} \vec{V}) \right]
\]

\( k - \) transport

\[
\frac{\partial}{\partial t} (\bar{p}k) + \frac{\partial}{\partial x} (\bar{p} \dot{u} \ddot{k}) + \frac{\partial}{\partial y} (\bar{p} \dot{v} \ddot{k}) - \frac{\partial}{\partial x} \left( \frac{\mu_{eff}}{\sigma_k} \frac{\partial k}{\partial x} \right) - \frac{\partial}{\partial y} \left( \frac{\mu_{eff}}{\sigma_k} \frac{\partial k}{\partial y} \right) = G - C_D \bar{p} \varepsilon
\]
\[ \varepsilon - \text{transport} \]
\[ \frac{\partial}{\partial t}(\bar{\rho}\varepsilon) + \frac{\partial}{\partial x}(\bar{\rho}\bar{u}\varepsilon) + \frac{\partial}{\partial y}(\bar{\rho}\bar{v}\varepsilon) - \frac{\partial}{\partial x} \left( \frac{\mu_{\text{eff}}}{\sigma_{\varepsilon}} \frac{\partial \varepsilon}{\partial x} \right) - \frac{\partial}{\partial y} \left( \frac{\mu_{\text{eff}}}{\sigma_{\varepsilon}} \frac{\partial \varepsilon}{\partial y} \right) = \left( C_{1E} G - C_{2} \bar{\rho} \varepsilon^{2} \right) / k \]

\[ H - \text{transport} \]
\[ \frac{\partial}{\partial t}(\bar{\rho}\bar{H}) + \frac{\partial}{\partial x}(\bar{\rho}\bar{u}\bar{H}) + \frac{\partial}{\partial y}(\bar{\rho}\bar{v}\bar{H}) - \frac{\partial}{\partial x} \left[ \left( \frac{\mu}{Pr} + \frac{\mu_{t}}{Pr_{t}} \right) \frac{\partial \bar{H}}{\partial x} \right] - \frac{\partial}{\partial y} \left[ \left( \frac{\mu}{Pr} + \frac{\mu_{t}}{Pr_{t}} \right) \frac{\partial \bar{H}}{\partial y} \right] = \]
\[ \frac{\partial}{\partial x} \left[ (\mu_{\text{eff}} - \Gamma_{H}) \frac{\partial V^{2}/2}{\partial x} - \Gamma_{H} \frac{\partial k}{\partial x} + \mu_{\text{eff}} \left( \frac{\partial \bar{u}^{2}/2}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} - \frac{2}{3} \bar{u} \text{div} \bar{V} \right) - \frac{2}{3} \bar{\rho} \bar{u} \bar{k} \right] + \frac{\partial}{\partial y} \left[ (\mu_{\text{eff}} - \Gamma_{H}) \frac{\partial V^{2}/2}{\partial y} - \Gamma_{H} \frac{\partial k}{\partial y} + \mu_{\text{eff}} \left( \frac{\partial \bar{v}^{2}/2}{\partial y} + \bar{u} \frac{\partial \bar{v}}{\partial x} - \frac{2}{3} \bar{v} \text{div} \bar{V} \right) - \frac{2}{3} \bar{\rho} \bar{v} \bar{k} \right] \]

\[ f - \text{transport} \]
\[ \frac{\partial}{\partial t}(\bar{\rho}f) + \frac{\partial}{\partial x}(\bar{\rho}\bar{u}f) + \frac{\partial}{\partial y}(\bar{\rho}\bar{v}f) - \frac{\partial}{\partial x} \left( \frac{\mu_{\text{eff}}}{\sigma_{f}} \frac{\partial f}{\partial x} \right) - \frac{\partial}{\partial y} \left( \frac{\mu_{\text{eff}}}{\sigma_{f}} \frac{\partial f}{\partial y} \right) = 0 \]

\[ m_{fu} - \text{transport} \]
\[ \frac{\partial}{\partial t}(\bar{\rho}m_{fu}) + \frac{\partial}{\partial x}(\bar{\rho}\bar{u}m_{fu}) + \frac{\partial}{\partial y}(\bar{\rho}\bar{v}m_{fu}) - \frac{\partial}{\partial x} \left( \frac{\mu_{\text{eff}}}{\sigma_{fu}} \frac{\partial m_{fu}}{\partial x} \right) - \frac{\partial}{\partial y} \left( \frac{\mu_{\text{eff}}}{\sigma_{fu}} \frac{\partial m_{fu}}{\partial y} \right) = \]
\[ \min \left[ R_{ED}, R_{GK} \right] \]

where
\[ V^{2} = \bar{u}^{2} + \bar{v}^{2} \]
\[ \mu_{\text{eff}} = \mu + \mu_{t} \]
\[ \text{div} \bar{V} = \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \]
\[ G = \mu_{\text{eff}} \left\{ 2 \left[ \left( \frac{\partial u}{\partial x} \right)^{2} + \left( \frac{\partial v}{\partial y} \right)^{2} \right] + \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^{2} \right\} \]
\[ - \frac{2}{3} \text{div} \bar{V} \cdot (\rho \bar{k} + \mu_{\text{eff}} \text{div} \bar{V}) \]

145
$$\bar{R}_{ED} = C \bar{p} \frac{\varepsilon}{k} \min \left[ \frac{\bar{m}_{fu}}{r_1}, \frac{\bar{m}_{O_2}}{r_1} \right]$$

$$\bar{R}_{GK} = A \bar{m}_{fu} \bar{m}_{O_2} \bar{p}^2 \exp \left[ - \frac{E}{(R T)} \right]$$

$$r_1 = \frac{\text{mass of } O_2}{\text{mass of fuel}}$$
Table 1

Constants in $k - \varepsilon$ model

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_1$</td>
<td>1.44</td>
</tr>
<tr>
<td>$C_2$</td>
<td>1.92</td>
</tr>
<tr>
<td>$C_D$</td>
<td>1.00</td>
</tr>
<tr>
<td>$C_\mu$</td>
<td>0.09</td>
</tr>
<tr>
<td>$\sigma_k$</td>
<td>1.0</td>
</tr>
<tr>
<td>$\sigma_\varepsilon$</td>
<td>$\frac{\kappa^2}{(C_2 - C_1) C_\mu^{1/2}}$</td>
</tr>
<tr>
<td>$E$</td>
<td>9.793</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>0.4187</td>
</tr>
</tbody>
</table>
## Table 2

Entries of $\phi$ and $R_k$

<table>
<thead>
<tr>
<th>$\phi$</th>
<th>$R_k$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$m_{fu}$</td>
<td>$R_{fu}$</td>
</tr>
<tr>
<td>$m_{O_2}$</td>
<td>$r_1 R_{fu}$</td>
</tr>
<tr>
<td>$m_{CO_2}$</td>
<td>$-r_2 R_{fu}$</td>
</tr>
<tr>
<td>$m_{H_2O}$</td>
<td>$-r_3 R_{fu}$</td>
</tr>
<tr>
<td>$\phi_A = m_{O_2} - r_1 m_{fu}$</td>
<td>0</td>
</tr>
<tr>
<td>$\phi_B = m_{CO_2} + r_2 m_{fu}$</td>
<td>0</td>
</tr>
<tr>
<td>$\phi_C = m_{H_2O} + r_3 m_{fu}$</td>
<td>0</td>
</tr>
</tbody>
</table>

where

\[ r_1 = \frac{\text{mass of } O_2}{\text{mass of fuel}} = \left(x + \frac{y}{4}\right) \frac{M_{O_2}}{M_{fu}} \]

\[ r_2 = \frac{\text{mass of } CO_2}{\text{mass of fuel}} = x \frac{M_{CO_2}}{M_{fu}} \]

\[ r_3 = \frac{\text{mass of } H_2O}{\text{mass of fuel}} = \frac{y}{2} \frac{M_{H_2O}}{M_{fu}} \]
Table 3

Governing Equations

\[
\frac{\partial}{\partial t} (\rho \phi) + \nabla \cdot (\rho u_i \phi - \Gamma_\phi \nabla \phi) = S_\phi
\]

<table>
<thead>
<tr>
<th>( \phi )</th>
<th>( \Gamma_\phi )</th>
<th>( S_\phi )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>( \mu_{\text{eff}} )</td>
<td>( \frac{\partial \bar{n}}{\partial x} + \frac{\partial}{\partial x} \left( \mu_{\text{eff}} \frac{\partial \bar{u}}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu_{\text{eff}} \frac{\partial \bar{v}}{\partial y} \right) - \frac{\partial}{\partial x} \left( \frac{2}{3} [\bar{p} + \mu_{\text{eff}} \text{div}\bar{V}] \right) )</td>
<td></td>
</tr>
<tr>
<td>( \mu_{\text{eff}} )</td>
<td>( - \frac{\partial \bar{n}}{\partial y} + \frac{\partial}{\partial x} \left( \mu_{\text{eff}} \frac{\partial \bar{u}}{\partial y} \right) + \frac{\partial}{\partial y} \left( \mu_{\text{eff}} \frac{\partial \bar{v}}{\partial y} \right) - \frac{\partial}{\partial y} \left( \frac{2}{3} [\bar{p} + \mu_{\text{eff}} \text{div}\bar{V}] \right) )</td>
<td></td>
</tr>
<tr>
<td>( k )</td>
<td>( \frac{\mu_{\text{eff}}}{\sigma_k} )</td>
<td>( G - C_D \bar{p} \varepsilon )</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>( \frac{\mu_{\text{eff}}}{\sigma_\epsilon} )</td>
<td>( (C_1 \varepsilon G - C_2 \bar{p} \varepsilon^2) / k )</td>
</tr>
<tr>
<td>( \bar{H} )</td>
<td>( \frac{\mu_{\text{eff}} - \Gamma_H}{\mu_{\text{eff}}} )</td>
<td>( \frac{\partial}{\partial x} \left[ \left( \frac{\partial \bar{V}^2/2}{\partial x} - \Gamma_H \frac{\partial k}{\partial x} + \mu_{\text{eff}} \left( \frac{\partial \bar{V}^2/2}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} - \frac{2}{3} \bar{u} \text{div}\bar{V} \right) \right] )</td>
</tr>
<tr>
<td>( \bar{m}_{f_u} )</td>
<td>( \frac{\mu_{\text{eff}}}{\sigma_{f_u}} )</td>
<td>( \min \left[ \bar{R}<em>{ED}, \bar{R}</em>{GK} \right] )</td>
</tr>
</tbody>
</table>

where

\[
\mu_{\text{eff}} = \mu + \mu_t
\]

\[
V^2 = \bar{u}^2 + \bar{v}^2
\]

\[
G = \mu_{\text{eff}} \left\{ 2 \left[ \frac{\partial \bar{u}}{\partial x} \right]^2 + \left( \frac{\partial \bar{v}}{\partial y} \right)^2 \right\} + \left( \frac{\partial \bar{u}}{\partial y} + \frac{\partial \bar{v}}{\partial x} \right)^2 \right\} - \frac{2}{3} \text{div}\bar{V} \cdot (\bar{p} + \mu_{\text{eff}} \text{div}\bar{V})
\]

\[
\bar{R}_{ED} = C \frac{\bar{p} \varepsilon}{k} \min \left[ \frac{\bar{m}_{f_u}}{r_1}, \frac{m_{O_2}}{r_1} \right]
\]

\[
\bar{R}_{GK} = A \frac{\bar{m}_{f_u} m_{O_2}}{\bar{p}^2} \exp \left[ - \frac{E}{(R T)} \right]
\]
Table 4

Under-relaxation Factors for One Chemical Species
(Cold Flow Without Fuel Injection)

<table>
<thead>
<tr>
<th>Variable</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \ddot{u} )</td>
<td>0.5</td>
</tr>
<tr>
<td>( \dot{v} )</td>
<td>0.5</td>
</tr>
<tr>
<td>( \ddot{p} )</td>
<td>1.0</td>
</tr>
<tr>
<td>( \dot{\rho} )</td>
<td>1.0</td>
</tr>
<tr>
<td>( k )</td>
<td>0.7</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>0.7</td>
</tr>
<tr>
<td>( \mu_{\text{eff}} )</td>
<td>0.7</td>
</tr>
<tr>
<td>( \tilde{H} )</td>
<td>0.7</td>
</tr>
<tr>
<td>( f )</td>
<td>1.0</td>
</tr>
<tr>
<td>( m_{fu} )</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Under-relaxation Factors for Multiple Chemical Species
(Cold Flow With Fuel Injection)

<table>
<thead>
<tr>
<th>Variable</th>
<th>Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \ddot{u} )</td>
<td>0.3</td>
</tr>
<tr>
<td>( \dot{v} )</td>
<td>0.3</td>
</tr>
<tr>
<td>( \ddot{p} )</td>
<td>0.8</td>
</tr>
<tr>
<td>( \dot{\rho} )</td>
<td>0.4</td>
</tr>
<tr>
<td>( k )</td>
<td>0.6</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>0.6</td>
</tr>
<tr>
<td>( \mu_{\text{eff}} )</td>
<td>0.6</td>
</tr>
<tr>
<td>( \tilde{H} )</td>
<td>0.7</td>
</tr>
<tr>
<td>( f )</td>
<td>0.7</td>
</tr>
<tr>
<td>( m_{fu} )</td>
<td>0.7</td>
</tr>
</tbody>
</table>
Figure 1. Schematic diagram of staggered grids:
(a) c cell; (b) u cell; (c) v cell.
Figure 2. Prescription of $\phi$ for western wall.
Figure 3. Boundary inlets: (a) north wall; (b) west wall.
Figure 4. Photograph of the sector-combustor rig.
Figure 5. Schematic diagram of the cross-section of the sector-combustor.
Figure 6. Schematic diagram of the air supply circuitry.
Figure 7. Principles of LDV: (a) Differential Doppler or dual-beam mode; (b) Fringe model.
Figure 8. Schematic diagram of LDV and data acquisition system.
Figure 9. Schematic diagram of seed generator.
Three Sections of the Combustor Investigated for the Cold Flow Without Fuel Injection

Figure 10. Schematic diagram of the three sections investigated in cold flow without fuel injection.
Figure 11. Velocity vectors in Section 1 of the cold flow without fuel injection.
Figure 12. Velocity vectors in Section 2 of the cold flow without fuel injection.
Figure 13. Velocity vectors in Section 3 of the cold flow without fuel injection.
Figure 14. Contours of the total velocity $V_t$ in Section 1 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection
Section 1 (Top-Jet Plane)

Total Velocity

Figure 15. Surface plot of the total velocity $V_t$ in Section 1 of the cold flow without fuel injection.
Figure 16. Contours of the total velocity $V_t$ in Section 2 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection
Section 2 (Inter-Jets Plane)

Total Velocity

Figure 17. Surface plot of the total velocity $V_t$ in Section 2 of the cold flow without fuel injection.
Figure 18. Contours of the total velocity $V_t$ in Section 3 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection

Section 3 (Bottom-Jet Plane)

Total Velocity

Figure 19. Surface plot of the total velocity $V_t$ in Section 3 of the cold flow without fuel injection.
Figure 20. Contours of the longitudinal velocity $\bar{u}$ in Section 1 of the cold flow without fuel injection
Cold Flow --- No Fuel Injection
Section 1 (Top-Jet Plane)

U velocity

Figure 21. Surface plot of the longitudinal velocity $\bar{u}$ in Section 1 of the cold flow without fuel injection
Figure 22. Contours of the longitudinal velocity $u$ in Section 2 of the cold flow without fuel injection.
Figure 23. Surface plot of the longitudinal velocity $\bar{u}$ in Section 2 of the cold flow without fuel injection.
Figure 24. Contours of the longitudinal velocity $\bar{u}$ in Section 3 of the cold flow without fuel injection.
Cold Flow — No Fuel Injection

Section 3 (Bottom-Jet Plane)

U velocity

Figure 25. Surface plot of the longitudinal velocity $\bar{u}$ in Section 3 of the cold flow without fuel injection.
Figure 26. Contours of the transverse velocity $\delta$ in Section 1 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection
Section 1 (Top-Jet Plane)

V velocity

Figure 27. Surface plot of the transverse velocity $\bar{v}$ in Section 1 of the cold flow without fuel injection.
Figure 28. Contours of the transverse velocity $\bar{v}$ in Section 2 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection
Section 2 (Inter-Jets Plane)

$V$ velocity

Figure 29. Surface plot of the transverse velocity $\tilde{v}$ in Section 2 of the cold flow without fuel injection.
Figure 30. Contours of the transverse velocity $\bar{v}$ in Section 3 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection
Section 3 (Bottom-Jet Plane)

V velocity

Figure 31. Surface plot of the transverse velocity \( \bar{v} \) in Section 3 of the cold flow without fuel injection.
Figure 32. Contours of the normal stress $u^2$ in Section 1 of the cold flow without fuel injection
Cold Flow --- No Fuel Injection
Section 1 (Top-Jet Plane)

Figure 33. Surface plot of the normal stress $u'^2$ in Section 1 of the cold flow without fuel injection.
Figure 34. Contours of the normal stress $\bar{u}^2$ in Section 2 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection
Section 2 (Inter-Jets Plane)

Figure 35. Surface plot of the normal stress $u'^2$ in Section 2 of the cold flow without fuel injection
Figure 36. Contours of the normal stress $u^{1/2}$ in Section 3 of the cold flow without fuel injection.
Figure 37. Surface plot of the normal stress $u'^2$ in Section 3 of the cold flow without fuel injection.
Figure 38. Contours of the normal stress $\sigma_{12}$ in Section 1 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection
Section 1 (Top-Jet Plane)

$\nu^2$

Figure 39. Surface plot of the normal stress $\nu^2$ in Section 1 of the cold flow without fuel injection.
EXPT. VNS

SECTION 2 (INTER-JETS PLANE)

Figure 40. Contours of the normal stress $\overline{v^2}$ in Section 2 of the cold flow without fuel injection.
Figure 41. Surface plot of the normal stress $\overline{v^2}$ in Section 2 of the cold flow without fuel injection.
EXPT. VNS

SECTION 3 (BOTTOM JET PLANE)

Figure 42. Contours of the normal stress $\sqrt{\nu}$ in Section 3 of the cold flow without fuel injection.
Figure 43. Surface plot of the normal stress $v^2$ in Section 3 of the cold flow without fuel injection.
Figure 44. Contours of the shear stress $u'^{}v'$ in Section 1 of the cold flow without fuel injection.
Figure 45. Surface plot of the shear stress $\overline{u'v'}$ in Section 1 of the cold flow without fuel injection.
Figure 46. Contours of the shear stress $\overline{u'u'}$ in Section 2 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection
Section 2 (Inter-Jets Plane)

\[ u'v' \]

Figure 47. Surface plot of the shear stress \( u'v' \) in Section 2 of the cold flow without fuel injection.
Figure 48. Contours of the shear stress $u'\nu'$ in Section 3 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection
Section 3 (Bottom-Jet Plane)

Figure 49. Surface plot of the shear stress $u'v'$ in Section 3 of the cold flow without fuel injection.
Figure 50. Contours of the shear stress intensity in Section 1 of the cold flow without fuel injection.
SECTION 2 (INTER-JETS PLANE)

Figure 51. Contours of the shear stress intensity in Section 2 of the cold flow without fuel injection.
Figure 52. Contours of the shear stress intensity in Section 3 of the cold flow without fuel injection.
Cuv

SECTION 1 (TOP JET PLANE)

Figure 53. Contours of the Reynolds stress correlation coefficient in Section 1 of the cold flow without fuel injection.
Cuv

SECTION 2 (INTER-JETS PLANE)

Figure 54. Contours of the Reynolds stress correlation coefficient in Section 2 of the cold flow without fuel injection
Figure 55. Contours of the Reynolds stress correlation coefficient in Section 3 of the cold flow without fuel injection.
Figure 56. Contours of the ratio of the normal stress intensities in Section 1 of the cold flow without fuel injection.
Figure 57. Contours of the ratio of the normal stress intensities in Section 2 of the cold flow without fuel injection.
Figure 58. Contours of the ratio of the normal stress intensities in Section 3 of the cold flow without fuel injection.
Figure 59. Contours of the turbulence kinetic energy $k$ in Section 1 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection
Section 1 (Top-Jet Plane)

Figure 60. Surface plot of the turbulence kinetic energy $k$ in Section 1 of the cold flow without fuel injection.
Figure 61. Contours of the turbulence kinetic energy $k$ in Section 2 of the cold flow without fuel injection.
Cold Flow --- No Fuel Injection
Section 2 (Inter-Jets Plane)

Figure 62. Surface plot of the turbulence kinetic energy $k$ in Section 2 of the cold flow without fuel injection.
Figure 63. Contours of the turbulence kinetic energy $k$ in Section 3 of the cold flow without fuel injection.
Cold Flow — No Fuel Injection

Section 3 (Bottom–Jet Plane)

$k$

Figure 64. Surface plot of the turbulence kinetic energy $k$ in Section 3 of the cold flow without fuel injection.
Figure 65. Contours of the ratio of $|u'v'|/k$ in Section 1 of the cold flow without fuel injection.
Figure 66. Contours of the ratio of $|u'v'|/k$ in Section 2 of the cold flow without fuel injection.
Figure 67. Contours of the ratio of $|\overline{uu'}|/k$ in Section 3 of the cold flow without fuel injection.
Figure 68. Contours of the turbulence intensity in Section 1 of the cold flow without fuel injection.
Figure 69. Contours of the turbulence intensity in Section 2 of the cold flow without fuel injection.
Figure 70. Contours of the turbulence intensity in Section 3 of the cold flow without fuel injection.
Cold Flow -- No Fuel Injection
Section 1 (Top-Jet Plane)

STREAKLINES

Figure 71. Streaklines in Section 1 of the cold flow without fuel injection.
Cold Flow -- No Fuel Injection
Section 2 (Inter-Jets Plane)

Figure 72. Streaklines in Section 2 of the cold flow without fuel injection.
Figure 73. Streaklines in Section 3 of the cold flow without fuel injection.
Figure 74. Photographs of the combustion flame:
(a) side view; (b) rear view.
Figure 75. Schematic diagram of the three sections investigated in the cold and hot flows with fuel injection.
Figure 76 (a). Velocity vector plot in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 76 (b). Velocity vector plot in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 77. Velocity vector plot in Section 2 of the hot flow.
Figure 78. Velocity vector plot in Section 3 of the hot flow.
Figure 79. The contour plot of the total velocity $V_t$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Cold Flow

Section 1 (Fuel Injector Plane)

Total Velocity

Figure 80. The surface plot of the total velocity $V_t$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 81. The contour plot of the total velocity $V_t$ in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 82. The surface plot of the total velocity $V_t$ in Section 1 (Fuel Injector Plane) of the hot flow.
Figure 8.3. The contour plot of the total velocity $V_t$ in Section 2 of the hot flow.
Figure 84. The surface plot of the total velocity $V_t$ in Section 2 of the hot flow.
Figure 85. The contour plot of the total velocity $V_t$ in Section 3 of the hot flow.
Figure 86. The surface plot of the total velocity $V_t$ in Section 3 of the hot flow.
Figure 87. The contour plot of the longitudinal velocity $\bar{u}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Cold Flow

Section 1 (Fuel Injector Plane)

U velocity

Figure 88. The surface plot of the longitudinal velocity $\bar{u}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 89. The contour plot of the longitudinal velocity $\bar{u}$ in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 90. The surface plot of the longitudinal velocity $\vec{u}$ in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 91. The contour plot of the longitudinal velocity $\bar{u}$ in Section 2 of the hot flow.
Figure 92. The surface plot of the longitudinal velocity $\bar{u}$ in Section 2 of the hot flow.
Figure 93. The contour plot of the longitudinal velocity \( \bar{u} \) in Section 3 of the hot flow.
Figure 94. The surface plot of the longitudinal velocity $u$ in Section 3 of the hot flow.
Figure 95. The contour plot of the transverse velocity \( \bar{\theta} \) in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Cold Flow
Section 1 (Fuel Injector Plane)

\( V \) velocity

Figure 96. The surface plot of the transverse velocity \( \tilde{v} \) in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 97. The contour plot of the transverse velocity $\bar{v}$ in Section 1 (Fuel-Injector Plane) of the hot flow.
Hot Flow

Section 1  (Fuel Injector Plane)

\( V \) velocity

Figure 98. The surface plot of the transverse velocity \( \vec{v} \) in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 99. The contour plot of the transverse velocity $\bar{v}$ in Section 2 of the hot flow.
Figure 100. The surface plot of the transverse velocity $\bar{v}$ in Section 2 of the hot flow.
Figure 101. The contour plot of the transverse velocity $\hat{v}$ in Section 3 of the hot flow.
Hot Flow
Section 3

\textbf{V velocity}

Figure 102. The surface plot of the transverse velocity $\tilde{v}$ in Section 3 of the hot flow.
Figure 103. The contour plot of the normal stress $u'^2$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 10. The surface plot of the normal stress $u'^2$ in Section 1 (Fuel Injector Plane) of the cold flow with fuel injection.
Figure 105. The contour plot of the normal stress $\bar{u}^2$ in Section 1 (Fuel-Injector Plane) of the hot flow.
Hot Flow

Section 1 (Fuel Injector Plane)

$u'^2$

Figure 106. The surface plot of the normal stress $u'^2$ in Section 1 (Fuel Injector Plane) of the hot flow.
Figure 107. The contour plot of the normal stress $\bar{u}^2$ in Section 2 of the hot flow.
Figure 108. The surface plot of the normal stress $u'^2$ in Section 2 of the hot flow.
Figure 109. The contour plot of the normal stress $\overline{u^2}$ in Section 3 of the hot flow.
Figure 110. The surface plot of the normal stress $u'^2$ in Section 3 of the hot flow.
Figure 111. The contour plot of the normal stress $\sqrt{\overline{v}}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 112. The surface plot of the normal stress $v'^2$ in Section 1 (FuelInjector Plane) of the cold flow with fuel injection.
Figure 113. The contour plot of the normal stress $\sigma_{zz}$ in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 114. The surface plot of the normal stress $\overline{v^2}$ in Section 1 (Fuel Injector Plane) of the hot flow.
Figure 115. The contour plot of the normal stress $\overline{v^2}$ in Section 2 of the hot flow.
Figure 116. The surface plot of the normal stress $\nu^2$ in Section 2 of the hot flow.
Figure 117. The contour plot of the normal stress $\sigma$ in Section 3 of the hot flow.
Figure 118. The surface plot of the normal stress $v'^2$ in Section 3 of the hot flow.
Figure 119. The contour plot of the shear stress $\overline{u'v'}$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 120. The surface plot of the shear stress $u'v'$ in Section 1 (Fuel Injector Plane) of the cold flow with fuel injection.
Figure 121. The contour plot of the shear stress $\bar{u}v'$ in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 122. The surface plot of the shear stress $u'v'$ in Section 1 (Fuel Injector Plane) of the hot flow.
Figure 123. The contour plot of the shear stress $\overline{u'v'}$ in Section 2 of the hot flow.
Hot Flow
Section 2
$u'v'$

Figure 124. The surface plot of the shear stress $u'v'$ in Section 2 of the hot flow.
EXPT. SS

HOT FLOW - SECTION 3

Figure 125. The contour plot of the shear stress $\overline{u'v'}$ in Section 3 of the hot flow.
Figure 126. The surface plot of the shear stress $u'v'$ in Section 3 of the hot flow.
Figure 127. The contour plot of the shear stress intensity in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 128. The contour plot of the shear stress intensity in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 129. The contour plot of the shear stress intensity in Section 2 of the hot flow.
Figure 130. The contour plot of the shear stress intensity in Section 3 of the hot flow.
Figure 131. The contour plot of the Reynolds stress correlation coefficient in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 132. The contour plot of the Reynolds stress correlation coefficient in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 133. The contour plot of the Reynolds stress correlation coefficient in Section 2 of the hot flow.
Figure 134. The contour plot of the Reynolds stress correlation coefficient in Section 3 of the hot flow.
Figure 135. The contour plot of the ratio of the normal stress intensities in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 136. The contour plot of the ratio of the normal stress intensities in Section 1 (Fuel-Injector Plane) of the hot flow with fuel injection.
Figure 137. The contour plot of the ratio of the normal stress intensities in Section 2 of the hot flow.
Figure 138. The contour plot of the ratio of the normal stress intensities in Section 3 of the hot flow.
Figure 139. The contour plot of the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 140. The surface plot of the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 141. The contour plot of the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 142. The surface plot of the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 143. The contour plot of the turbulence kinetic energy $k$ in Section 2 of the hot flow.
Figure 144. The surface plot of the turbulence kinetic energy $k$ in Section 2 of the hot flow.
Figure 145. The contour plot of the turbulence kinetic energy $k$ in Section 3 of the hot flow.
Figure 146. The surface plot of the turbulence kinetic energy $k$ in Section 3 of the hot flow.
Figure 147. The contour plot of the ratio of the shear stress $\overline{u'v'}$ and the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 148. The contour plot of the ratio of the shear stress $\overline{u'v'}$ and the turbulence kinetic energy $k$ in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 149. The contour plot of the ratio of the shear stress $\overline{u'v'}$ and the turbulence kinetic energy $k$ in Section 2 of the hot flow.
Figure 150. The contour plot of the ratio of the shear stress $\overline{u'v'}$ and the turbulence kinetic energy $k$ in Section 3 of the hot flow.
Figure 151. The contour plot of the turbulence intensity in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 152. The contour plot of the turbulence intensity in Section 1 (Fuel-Injector Plane) of the hot flow.
Figure 153. The contour plot of the turbulence intensity in Section 2 of the hot flow.
Figure 154. The contour plot of the turbulence intensity in Section 3 of the hot flow.
Figure 155. Comparison of the mean velocity \( \bar{u} \) between the hot and cold flows.
Figure 156. Comparison of the transverse mean velocity $\bar{v}$ between the hot and cold flows.
Figure 157. Comparison of the turbulence kinetic energy $k$ between the hot and cold flows.
Figure 158. Comparison of the shear stress $\overline{u'v'}$ between the hot and cold flows.
Cold Flow With Fuel Injection

Section 1

STREAKLINES

Figure 159. Streaklines in Section 1 (Fuel-Injector Plane) of the cold flow with fuel injection.
Figure 160. Streaklines in Section 1 (Fuel-Injector Plane) of the hot flow.
Hot Flow
Section 2
STREAKLINES

Figure 161. Streaklines in Section 2 (Fuel-Injector Plane) of the hot flow.
Figure 162. Streaklines in Section 3 (Fuel-Injector Plane) of the hot flow.
Figure 163. The computed streamline in Section 1 (Top-Jet Plane) of the cold flow without fuel injection.
Figure 164. The computed streamline in Section 2 (Inter-Jets Plane) of the cold flow without fuel injection.
Figure 16.5. The computed streamline in Section 3 (Bottom-Jet Plane) of the cold flow without fuel injection.
Figure 166. The computed streamline in the Hypothetical Plane of the cold flow without fuel injection.
Figure 167. Comparison of the measured and the predicted longitudinal velocity $\bar{u}$ for cold flow without fuel injection.
Figure 168. Comparison of the measured and the predicted transverse velocity $\bar{v}$ for cold flow without fuel injection.
Figure 169. Comparison of the measured and the predicted turbulence kinetic energy $k$ for cold flow without fuel injection
Figure 170. Comparison of the measured and the predicted shear stress $u'v'$ for cold flow without fuel injection.
Figure 171. The computed streamline in Section 1 (Top-Jet Plane) of the cold flow with fuel injection.
Figure 172. The total velocity $V_t$ of the predicted flow in Section 1 (Top-Jet Plane) of the cold flow with fuel injection.
Figure 173. The mixture fraction $f$ of the predicted flow in Section 1 (Top-Jet Plane) of the cold flow with fuel injection.
Figure 174. Velocity vectors in Section 1 measured by LDV and predicted by Lai (1990) in cold flow without fuel injection.
Figure 175. Velocity vectors in Section 2 measured by LDV and predicted by Lai (1990) in cold flow without fuel injection.
Figure 176. Velocity vectors in Section 3 measured by LDV and predicted by Lai (1990) in cold flow without fuel injection.
An experimental and computational investigation of an annular reverse-flow combustor

Hu, T.C.J., Cusworth, R.A., Sislian, J.P.

The complex flowfield of a Pratt & Whitney Canada toroidal vortex annular reverse-flow sector-combustor has been investigated. The u and v components of the mean velocity and the corresponding turbulence intensities $u'$, $v'$ and $w'$ were measured in detail for cold flow with and without fuel injection and hot flow conditions, with a two-component argon-ion laser Doppler velocimeter operated in dual-beam, forward scatter mode. The flowfield characteristics were identified from the LDV data and the flow visualization pictures substantiated the findings. Effects of heat addition on the flowfield were determined from comparisons of the cold and hot flows. Results show that combustion intensifies vortical and recirculating flows, increases the momentum of the fuel jet, locally laminarizes the fuel jet near the injector inlet, and increases the turbulence kinetic energy and turbulent stresses at the shear layers. The combustor aerodynamic developments in the flowfield with and without fuel injection are discussed. Predictions of the cold flow using a 2-D TEACH-type computer code demonstrated that the code can provide qualitative agreement for flow regions which are not strongly 3-D in nature, whereas a 3-D numerical model is required in resolving all the flow features realistically. Researchers and engineers will find the measured data essential in the understanding, evaluations and developments of combustor designs, and mathematical modeling of processes inside practical combustors.

A limited number of copies of this report are printed for distribution free of charge. Photocopies at $15.00 (Canadian) will be supplied on request when printed copies are no longer available.

University of Toronto, Institute for Aerospace Studies (UTIAS)
4925 Dufferin Street, Downsview, Ontario, Canada, M3H 5T6

UTIAS Report No. 338

AN EXPERIMENTAL AND COMPUTATIONAL INVESTIGATION OF AN ANNULAR REVERSE-FLOW COMBUSTOR

Hu, T.C.J., Cusworth, R.A., Sislian, J.P.