*Combustion and Propulsion Research Laboratory
Mechanical and Industrial Engineering Department
The University of Texas at El Paso
El Paso, TX 79968*

**Final Technical Report on
Department of Energy Grant DE-FG26-01NT41363**

Entitled

**Passive Control of Particle Dispersion in a Particle-laden Circular Jet using
Elliptic Co-annular Flow: A means for Improving Utilization and
Emission Reductions in Pulverized Coal Burner.**

**Author
Ahsan R. Choudhuri, PhD**

June 2003

Disclaimer

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

Project Information

Project Title: Passive Control of Particle Dispersion in a Particle-laden Circular Jet using Elliptic Co-annular Flow: A means for Improving Utilization and Emission Reductions in Pulverized Coal Burner

Grant No: DE-FG26-01NT41363

Agency: National Energy Technology Laboratory
Department of Energy

Program: Topic 9: Faculty/Student Exploratory Research Training Grant Program (Support of Advanced Fossil Resource Utilization Research by Historically Black Colleges and Universities and Other Minority Institutions)

DOE Project Manager: Bruce W. Lani
Project Manager/NOx Coordinator
Environmental Projects Division
U.S. Department of Energy
National Energy Technology Laboratory
626 Cochran Mill Rd., MS-922-273C, P.O. Box 10940
Pittsburgh, PA 15236
Tel: (412) 386-5819, Fax: 412-386-5917
E-mail: lanl@netl.doe.gov

Amount: \$20,000

Project Period: 9/30/2001-09/29/2002

Principal Investigator: Ahsan R. Choudhuri, PhD
Assistant Professor

Contact Address: Department of Mechanical and Industrial Engineering
The University of Texas at El Paso
500 West University, Mail Stop: 0521, El Paso, Texas 79968
Tel: 915 747 6905, Fax: 915 747 5019, E-mail: ahsan@utep.edu

Students: (i) Abel Vargas (ii) Fernando Jasso

Preface

This is the final technical report on DOE Faculty/Student Exploratory Research Training Grant DE-FG26-01NT41363 entitled “Passive Control of Particle Dispersion in a Particle-laden Circular Jet using Elliptic Co-annular Flow: A means for Improving Utilization and Emission Reductions in Pulverized Coal Burner”. Technical work supported by this grant is described in detail in two Master of Science theses by Abel Vargas (completed) and Fernando Jasso (in preparation).

Abstract

A passive control technology utilizing elliptic co-flow to control the particle flinging and particle dispersion in a particle (coal)-laden flow was investigated using experimental and numerical techniques. Preferential concentration of particles occurs in particle-laden jets used in pulverized coal burner and causes uncontrollable NO_x formation due to inhomogeneous local stoichiometry. This particular project was aimed at characterizing the near-field flow behavior of elliptic coaxial jets. The knowledge gained from the project will serve as the basis of further investigation on fluid-particle interactions in an asymmetric coaxial jet flow-field and thus is important to improve the design of pulverized coal burners where non-homogeneity of particle concentration causes increased NO_x formation.

Executive Summary

Coaxial jets, in which an annular jet surrounds a central jet, arise in numerous combustion applications due to their vigorous mixing characteristics. The mixing is dominated by the vortex rings whose interactions govern the growth, entrainment, and mixing of the jet. Since the conversion of nitrogen containing species to NO_x strongly depends on the local stoichiometry, the introduction of longitudinal vortices into the inner jet shear layer to control the particle dispersion in a pulverized coal burner is one potential passive NO_x control techniques. Elliptic jets in which longitudinal vortices are formed due to asymmetric self-induction of the vortex ring structures can be effectively used to change the particle dispersion characteristics of the particle-laden jet. However, taking an aerodynamic path for NO_x control requires not only an understanding of particle dynamics but an equal understating of single phase flow-field. The particle-fluid interactions are very complex in nature, so a complete understanding of the instantaneous and mean velocity field is necessary prior to the tracking of individual particle path. Although the overall research goal is to explore the potential of using elliptic co-flow to control the particle flinging and preferential particle concentration, this particular project was aimed at quantifying the near-field flow characteristics of elliptic coaxial jets. The information obtained from the project will serve as the basis of further investigation on fluid-particle interaction and thus is important to improve the design of pulverized coal burners where non-homogeneity of particle concentration causes increased NO_x formation.

To understand the effects of a elliptic coflow on a circular inner jet flow-field, the near-field flow characteristics of a turbulent elliptical coaxial jet with 0.55 and 1.45 velocity ratios ($m = U_o/U_i$) were numerically computed and experimentally measured. To design an elliptic coaxial flow nozzle, flows through two-dimensional and three-dimensional coflow nozzles were also calculated. The flow characteristics were compared with a baseline condition (circular coflow).

CAD (Computer Aided Drawing) models of two-dimensional annuli, a three-dimensional circular and elliptical coaxial nozzle, and two jet domains were developed to capture the flow behavior in all geometries. The geometries in each simulation consisted of structure grids, with fluid properties obtained at ambient conditions.

Numerical studies were then conducted on laminar flow through two-dimensional annuli of different outer and inner wall contours and lengths. The understanding of the flow within the two-dimensional annulus led to the development of a three-dimensional circular and elliptical coaxial nozzle. The coaxial nozzles were restricted to an inlet radius of 9.649 cm (3.799 inches) and an exit area of 17.129 cm² (2.655 in²), with the elliptical nozzle having 2:1 aspect ratio. An investigation of flow inside the three-dimensional annulus of the coaxial nozzles was then conducted. The final step included numerically computed near-field flow characteristics of circular and elliptic coaxial jets at different velocity ratios.

Numeric results for the 2-D annuli indicated uniformity in exit velocity as the length of the annulus was increased. Also, a 5th order polynomial outer wall contour produces a faster acceleration inside the annulus when compared to a 4th and 3rd order outer wall profile. Flow in the annulus of the three-dimensional coaxial nozzles exhibit an increase in the downstream direction with no abrupt changes in velocity. An annular exit velocity of approximately 22.0 m/s was obtained for the circular and elliptical coaxial nozzle.

Elliptical coflow analyses reveal the jet approaching symmetry at $x = 30.48$ cm. The first half of axis switching is captured in the numeric simulations. A dependence on the outer structures with different velocity ratios is observed in the elliptical coflow jet. Radial u velocity plots show a wider jet produced on the major axis plane. Merging of the inner and outer flow occurs earlier with a velocity ratio of 0.55. The coflow analyses show a higher concentration of turbulent kinetic energy in the outer shear layer. The rms axial velocity is minimal, less than 0.5 m/s at the center of the jet up to $x = 25.4$ cm for

both velocity ratios, and increases away from the centerline with the maximum value where the outer shear layer forms.

The elliptic coaxial nozzle shear layers exhibit very distinct spatial flow features which will certainly have a profound impact on the fluid-particle interaction characteristics. For instance, the wider shear layer growth in the major axis plane causes rapid spreading of fluid and highly asymmetric local flow structure, which may eliminate the nonhomogeneity of local fuel stoichiometry and thus reduce the NO_x formation. Similarly, the elliptic coaxial jet injects longitudinal vortices inside inner shear layer through asymmetric self-induction which causes higher mass entrainment and thus the centerline velocity to decay faster (for $m = 0.55$) than the circular coaxial nozzle. Also, as it is observed from the spreading jet contour plots, the increase of mass inside the jet results from the pumping mechanism (axis switching) caused by the contraction of the major axis and the expansion of the minor axis. This flow feature of elliptic coaxial jet can also be used to generate a fuel-rich/fuel-lean sequence in near burner region to reduce the NO_x formation.

Table of Contents

NOMENCLATURE	XII
LIST OF TABLES	XV
LIST OF FIGURES	XVI
CHAPTER 1	1
INTRODUCTION	1
1.1 OBJECTIVES	3
1.2 PROJECT OVERVIEW	3
1.3 ORGANIZATION OF THE REPORT	4
CHAPTER 2	6
LITERATURE REVIEW	6
2.1 JET THEORY	6
2.1.1 Non-Reacting Jets	9
2.1.2 Reacting Jets	11
2.2 ASYMMETRIC JETS	12
2.2.1 Asymmetric Reacting Jets	14
2.4 COFLOW	20
CHAPTER 3	26
NUMERICAL METHODS	26
3.1 GOVERNING EQUATIONS	26
3.1.1 Continuity Equation	27
3.1.2 Momentum Equation	29
3.1.3 Energy Equation	32
3.2 TURBULENT MODELING	34
3.3 DISCRETIZATION METHOD	38
3.3.1 Convective Discretization	40
3.3.1.1 Upwind Scheme	41
3.3.2 Diffusion Discretization	42
3.3.3 Source Term Linearization	43
3.3.4 Finite Difference Equation	44
3.4 SOLUTION ALGORITHM	44
CHAPTER 4	48
COMPUTATIONAL METHODOLOGY	48
4.1 ANNULAR DESIGN	49

4.1.1 Geometry Generation – Annular Design	49
4.1.2 Numerical Modeling – Annular Design	52
4.2 COAXIAL NOZZLE DESIGN	54
4.2.1 Geometry Generation – Coaxial Nozzle Design	54
4.2.2 Numerical Modeling – Coaxial Nozzle Design	60
4.3 COFLOW JET ANALYSIS	61
4.3.1 Geometry Generation – Coflow Jet Analysis	62
4.3.2 Numerical Modeling – Coflow Jet Analysis	66
CHAPTER 5	70
EXPERIMENTAL FACILITY AND METHODS	70
5.1 EXPERIMENTAL COMPONENTS	70
5.1.1 Coaxial Jet Rig	71
5.1.2 Traverse Mechanism	76
5.1.3 Data Acquisition and Instrumentation Control	77
5.2 EXPERIMENTAL INSTRUMENTATION	78
5.3 EXPERIMENTAL TECHNIQUES	80
CHAPTER 6	82
COMPUTATIONAL AND EXPERIMENTAL RESULTS	82
SECTION 6-A	83
6-A.1 TWO DIMENSIONAL ANNULUS	83
6-A.1.1 Effect of Curvature	83
6-A.1.2 Effect of Length	89
6-A.2 THREE-DIMENSIONAL CIRCULAR AND ELLIPTICAL COAXIAL JETS	92
6-A.3 VALIDATION OF NUMERIC RESULTS	96
SECTION 6-B	97
6-B.1 VELOCITY MAGNITUDE CONTOUR PLANES	97
6-B.1.1 Contour Planes for $m = 0.55$	98
6-B.1.2 Contour Planes for $m = 1.45$	103
6-B.1.3 Comparison Between $m = 0.55$ and $m = 1.45$	108
6-B.2 VORTICITY CIRCULATION CONTOUR PLANES	112
6-B.2.1 Contour Planes for $m = 0.55$	113
6-B.2.2 Contour Planes for $m = 1.45$	118
6-B.2.3 Comparison Between $m = 0.55$ and $m = 1.45$	123
6-B.3 SPREADING JET CONTOURS	123
6-B.3.1 Spreading Jet Contours for $m = 0.55$	124
6-B.3.2 Spreading Jet Contour for $m = 1.45$	124
6-B.3.3 Comparison Between $m = 0.55$ and $m = 1.45$	125
6-B.4 CENTERLINE VELOCITY AND TURBULENT KINETIC ENERGY	130
6-B.4.1 Centerline Velocity for $m = 0.55$	130

6-B.4.2 Centerline Velocity for $m = 1.45$	131
6-B.4.3 Turbulent Kinetic Energy	137
6-B.5 Radial Velocity Distribution.....	142
6-B.6 RADIAL TURBULENT KINETIC ENERGY	148
CHAPTER 7	158
CONCLUSIONS AND RECOMMENDATIONS	158
7.1 TWO-DIMENSIONAL ANNULUS	159
7.2 COAXIAL NOZZLE DESIGN	159
7.3 COFLOW JET ANALYSIS	160
7.4 PRACTICAL IMPLICATIONS	165
7.5 RECOMMENDATIONS FOR FUTURE WORK	165
7.6 RESEARCH INFRASTRUCTURE, RESEARCH TRAINING AND EDUCATION	166
BIBLIOGRAPHY	167
APPENDIX A – DERIVATION OF POLYNOMIAL PROFILE CURVES.....	172
APPENDIX B – CURVE IMPORTING STEPS	175
APPENDIX C – HOT WIRE PROBE CALIBRATION PROCEDURES	178

Nomenclature

English

a	Link coefficient
A	Area
A_i	Inlet area of the control volume
a_x	Acceleration in the x-direction
C	Constant
$C_{\varepsilon 1}$	Coefficient of production of ε in ε -equation
$C_{\varepsilon 2}$	Coefficient of destruction of ε in ε -equation
C_u	Coefficient in the eddy viscosity relation in k- ε model
C_v	Specific heat
D_e	Equivalent Diameter
d_i	Inner nozzle exit diameter
E	Energy of the system
E	Eastern node in the discretized control volume
e	Internal energy
F	Force
F_x	Force exerted in the x-direction
g	Gravitational Constant
h	Static enthalpy
J	Momentum flux
K	Velocity decay coefficient
K	Thermal Conductivity
k	Turbulent kinetic energy
l	Length scale
ℓ	Length
L	Characteristic length
m	Mass
m	Velocity ratio (U_o/U_i)
M	Molecular weight
m^{-1}	Mean velocity ratio (U_i/U_o)
m_f	Mass flux
$\dot{m}$	Mass flow rate
$\dot{m}_e$	Mass flow exiting the control volume
$\dot{m}_i$	Mass flow entering the control volume
N	Northern node in the control discretized volume
P	Pressure
P	Center node in the discretized control volume

P_e	Pressure at the exit of the control volume
P_i	Pressure at the inlet of the control volume
q	Heat Transfer
Q	Volumetric flow rate
Q_e	Volumetric flow rate exiting the control volume
Q_i	Volumetric flow rate exiting the control volume
r	Radius, radial coordinate
R	Universal gas constant
$R_{3.0}$	Polynomial function for 3.0 inch long annulus
$r_{3.5}$	Polynomial function for 3.5 inch long annulus
r_{3rd}	3 rd order polynomial function for the annulus' outer wall
$r_{4.0}$	Polynomial function for 4.0 inch long annulus
r_{4th}	4 th order polynomial function for the annulus' outer wall
r_{5th}	5 th order polynomial function for the annulus' outer wall
Re	Reynolds number
Re_i	Inner jet exit Reynolds number
Re_o	Outer jet exit Reynolds number
r_{in}	Radius of the inner nozzle
r_{maj}	Major axis radius
r_{min}	Minor axis radius
r_o	Exit radius of outer nozzle
S	Southern node in the control discretized volume
S	Source term
T	Temperature
t	Time
T_i	Turbulent intensity
u	Cartesian component of velocity in x -direction
u'	Root Mean Square (rms) velocity in the x -direction
$\tilde{u}$	Time averaged mean velocity
U_c	Centerline velocity
U_i	Inner nozzle exit velocity
U_o	Outer nozzle exit velocity
U_{ref}	Mean velocity in all three components
V	Velocity vector containing all three velocity components
$ \mathbf{V} $	Velocity magnitude
V	Volume
v	Cartesian component of velocity in y -direction
$\mathbf{v}'$	Velocity vector in a 2-D plane
$\vec{v}$	Velocity exiting the control volume in a 2-D plane
$\vec{v}_e$	Velocity entering the control volume in a 2-D plane
$\vec{v}_i$	Root Mean Square (rms) velocity in the y -direction
w	Cartesian component of velocity in z -direction

W	Western node in the discretized control volume
W	Work done on or by the system
w'	Root Mean Square (rms) velocity in the z-direction
x	Cartesian coordinate
x_p	Distanced measured from the exit of the jet to any point along the streamline
y	Cartesian coordinate

Greek

Ω	Angular velocity
ω	Vorticity
ρ	Density
μ	Molecular (laminar) dynamic viscosity
Γ	Diffusion coefficient
Γ_t	Turbulent diffusivity coefficient
ε	Rate of turbulence dissipation
ε_{res}	Residuals
ν	Kinematic viscosity
$\mathcal{V}$	Velocity scale
μ	Dynamic viscosity
η	Similarity variable
ϕ	General scalar flow variable
δ_{ij}	Kronekar Delta
τ	Shear stress
σ	Normal stress
μ_t	Eddy (turbulent) dynamic viscosity
∇	Vector operator
σ_t	Turbulent Prandtl number

List of Tables

Table 4-1. Properties of air obtained at $T = 20\text{ }^{\circ}\text{C}$ and $P = 101.325\text{ kPa}$	48
Table 4-2. Boundary and initial conditions applied to the 2-D numeric simulations.	53
Table 4-3. Boundary and initial conditions applied to the 3-D coaxial nozzle analysis.	61
Table 4-4. Boundary and initial conditions applied to the domains of the coflow jet simulations.	68
Table 5-1. Inner, circular, and elliptical nozzle specifications.	76
Table 5-2. Experimental equipment.	80
Table 5-3. Experimental conditions.	81
Table 6-1. Percent difference between velocities near the outer and inner wall of an annulus of varying length.	91
Table 6-2. Percent difference between the experimental and numerical centerline velocities when $m = 0.55$ for an axis- and non-axisymmetric coaxial nozzle.	131
Table 6-3. Percent difference between the experimental and numerical centerline velocities when $m = 1.45$ for an axis- and non-axisymmetric coaxial nozzle.	132

List of Figures

Figure 2-1. Different zones of a spreading jet (Malmstrom et al. 1997).	9
Figure 2-2. Top hat velocity distribution at the jet's exit (White 1990).	9
Figure 2-3. Evolution of a jet's instability with advancing Reynolds number. Parts a) – d) span the Reynolds number from around 10^2 to 10^3 (Crow & Champagne 1971).	10
Figure 2-4. Sketch of the development of axis switching (Hussain & Husain 1989).	13
Figure 2-5. A front and back view of axisymmetric coaxial jet.	17
Figure 2-6. Flow development in a coaxial jet (Champagne and Wygnanski 1971).	18
Figure 2-7. The three zones of a coaxial jet (Tang & Ko 1994).	19
Figure 2-8. Schematic of coflow on combustion.	21
Figure 2-9. Chemical Pathways of NO_x Formation and Destruction.	23
Figure 2-10. Low NO_x Burner Concept.	24
Figure 2-11. Low NO_x DRB-4Z TM Burner (Credit B & W).	25
Figure 3-1. Fluid element of volume $dx dy dz$ with mass fluxes through various faces (Anderson 1995).	28
Figure 3-2. Forces in the x-direction acting on the faces of fluid element (Anderson 1995).	30
Figure 3-3. Control volume for a two-dimensional grid.	39
Figure 3-4. Discretized domain used in the upwind scheme with flow entering the control volumes a) positive flow b) negative flow.	41
Figure 3-5. Discretized domain with control volume.	42
Figure 3-6. Solution flowchart for the SIMPLEC algorithm.	47
Figure 4-1. a) Computational domain for the two dimensional annulus. b) Meshed	52

Figure 4-2. Wire frame of the circular coaxial nozzle generated by a 45° revolution of the four inch annulus.....	55
Figure 4-3. Surfaces needed to construct the 3D mesh on the top half of the coaxial nozzle.....	56
Figure 4-4. a) Top half meshed nozzle. b) Complete 3-D mesh.....	56
Figure 4-5. Wire frame of the elliptical nozzle.....	58
Figure 4-6. a) Example of rotation. b) Example of revolution	59
Figure 4-7. a) Three-dimensional mesh on a quarter of the non-axisymmetric coaxial nozzle. b) Complete 3-D mesh on the non-axisymmetric coaxial nozzle.....	60
Figure 4-8. a) Dimension of the circular domain. b) Circular domain with extruded exit jet profiles and horizontal planes needed to create a total of six surfaces.....	63
Figure 4-9. Three-dimensional meshed domain used to capture the jet produced by the axisymmetric coaxial nozzle.	64
Figure 4-10. a) Wire frame of the elliptical domain. b) Three-dimensional meshed domain used to capture the jet produced by the elliptical coaxial nozzle.....	66
Figure 5-1. Coaxial jet test rig support frame,	71
Figure 5-2. Schematic of coaxial jet flow system.....	72
Figure 5-3. Air lines and components that supply air to the inner and outer nozzle.	73
Figure 5-4. Aluminum 7.62 cm long inner nozzle with 5th order contraction.	74
Figure 5-5. Coaxial jet test facility and enlarged annular chamber.	75
Figure 5-6. a) Circular nozzle with a 5.08 cm diameter. b) Elliptical nozzle with a 2:1 aspect ratio.....	76
Figure 5-7. Data acquisition and instrumentation control set up.	77
Figure 5-8. TSI 1210-20 tungsten hot-wire probe sensor (www.tsi.com).....	78

Figure 6-1. Velocity contours within the annuli of fixed inner wall and varying outer wall.	85
Figure 6-2. Axial velocities at three different radial locations measured from the inner wall of an annulus.	86
Figure 6-3. Radial u velocity distributions at axial locations of 5.08 cm and 7.62 cm.	88
Figure 6-4. Radial u velocity distribution at an axial location of 10.16 cm.	89
Figure 6-5. Velocity contours within the annuli with varying length.	90
Figure 6-6. Exit velocity for annuli of varying length.	91
Figure 6-7. Velocity contours within the annulus of a coaxial nozzle with a) circular and b) elliptical outer nozzle.	92
Figure 6-8. Annular u velocity at three different radial locations measured from the inner wall of the circular or elliptical coaxial nozzles.	93
Figure 6-9. Outer wall profiles for the circular, elliptical, and inner nozzles.	94
Figure 6-10. Radial velocity distributions at three different axial locations.	95
Figure 6-11. Magnitude velocity contours on the circular XY planes for $m = 0.55$	100
Figure 6-12. Magnitude velocity contours on the minor axis plane for $m = 0.55$	101
Figure 6-13. Magnitude velocity contours on the major axis planes for $m = 0.55$	102
Figure 6-14. Magnitude velocity contours on the circular XY planes for $m = 1.45$	105
Figure 6-15. Magnitude velocity contours on the minor axis plane for $m = 1.45$	106
Figure 6-16. Magnitude velocity contour on the minor axis plane for $m = 1.45$	107
Figure 6-17. Global magnitude velocity contours on the circular XY plane for $m = 1.45$	109
Figure 6-18. Global magnitude velocity contours on the minor axis plane for $m = 1.45$	110
Figure 6-19. Global magnitude velocity contours on the major axis plane for $m = 1.45$	111

Figure 6-20. Vorticity circulation contours on the circular XY plane for $m = 0.55$	115
Figure 6-21. Vorticity circulation contours on the minor axis plane for $m = 0.55$	116
Figure 6-22. Vorticity circulation contours on the major axis plane for $m = 0.55$	117
Figure 6-23. Vorticity circulation contours on the circular XY plane for $m = 1.45$	120
Figure 6-24. Vorticity circulation contours on the minor axis plane for $m = 1.45$	121
Figure 6-25. Vorticity circulation contours on the major axis plane for $m = 1.45$	122
Figure 6-26. Spreading jet velocity contours of an axisymmetric and non-axisymmetric coaxial nozzle at various axial distances when $m = 0.55$	126
Figure 6-27. Continued spreading jet velocity contours of a axisymmetric and non-axisymmetric coaxial nozzle at various axial distances when $m = 0.55$	127
Figure 6-28. Spreading jet velocity contours of an axisymmetric and non-axisymmetric coaxial nozzle at various axial distances when $m = 1.45$	128
Figure 6-29. Continued spreading jet velocity contours of an axisymmetric and non-axisymmetric coaxial nozzle at various axial distances when $m = 1.45$	129
Figure 6-30. Experimental and computational centerline velocity at $m = 0.55$	133
Figure 6-31. Dimensionless experimental and computational centerline velocity at $m = 0.55$	134
Figure 6-32. Experimental and computational centerline velocity at $m = 1.45$	135
Figure 6-33. Dimensionless experimental and computational centerline velocity at $m = 1.45$	136
Figure 6-34. Centerline turbulent kinetic energy for an axisymmetric and non-axisymmetric coaxial nozzle for $m = 0.55$	138
Figure 6-35. Dimensionless centerline turbulent kinetic energy for an axisymmetric and non-axisymmetric coaxial nozzle for $m = 0.55$	139

Figure 6-36. Centerline turbulent kinetic energy for an axisymmetric and non-axisymmetric coaxial nozzle for $m = 1.45$.	140
Figure 6-37. Dimensionless centerline turbulent kinetic energy for an axisymmetric and non-axisymmetric coaxial nozzle for $m = 1.45$.	141
Figure 6-38. Evolution of radial u velocity exiting from the circular coaxial nozzle with $m = 0.55$.	143
Figure 6-39. Radial velocity distributions in the circular XY, minor axis, and major axis planes for velocity ratio of 0.55.	144
Figure 6-40. Dimensionless radial velocity distributions for $m = 0.55$.	145
Figure 6-41. Radial velocity distributions in the circular XY, minor axis, and major axis planes for velocity ratio of 1.45.	146
Figure 6-42. Dimensionless radial velocity distributions for $m = 1.45$.	147
Figure 6-43. Turbulent kinetic energy radial distribution plots in the circular XY, minor axis, and major axis planes for velocity ratio of 0.55.	150
Figure 6-44. Dimensionless turbulent kinetic energy radial plots for $m = 0.55$.	151
Figure 6-45. Turbulent kinetic energy radial distribution plots in the circular XY, minor axis, and major axis planes for velocity ratio of 1.45.	152
Figure 6-46. Dimensionless turbulent kinetic energy radial plots for $m = 1.45$.	153
Figure 6-47. u fluctuating velocity radial distribution plots in the circular XY, minor axis, and major axis planes for velocity ratio of 0.55.	154
Figure 6-48. Dimensionless u' radial plots for $m = 0.55$.	155
Figure 6-49. u fluctuating velocity radial distribution plots in the circular XY, minor axis, and major axis planes for velocity ratio of 1.45.	156
Figure 6-50. Dimensionless u' radial plots for $m = 1.45$.	157

Chapter 1

INTRODUCTION

Pulverized coal combustion is widely used in utility boilers, industrial boilers, furnace, kilns, and other energy conversion devices. Clean utilization of pulverized coal is a major problem in combustion processes especially in those burning low-grade fuels. To reach low pollutants emission level and high combustion efficiency in a pulverized coal combustor, a favorable aerodynamic field has to be arranged and particle behaviors have to be properly controlled to match the flow pattern. Proper aerodynamic field and particle behaviors are also crucial for achieving on-time ignition and flame stabilization in a pulverized coal flame.

In pulverized coal suspensions, the local stoichiometry is determined by the extent of particle dispersion. Hence, particle behavior and their interactions with the recirculating flow have pronounced effects on the combustion and emission performance of the combustor. Recent studies that address the effect of large-scale coherent structures on particle transport and dispersion in non-reacting jets and mixing layers show that particle clusters and voids may be formed as a result of the interaction of particles with vortical structures (Wicker and Eaton, 2001; Wark et al., 2000, Park et al., 1998, Wicker and Eaton, 1994; Longmire and Eaton, 1992). The complex particle-vortex interactions cause particles to be flung outward on the downstream side of each vortex ring (Wicker and Eaton, 2001; Wicker and Eaton, 1994; Longmire and Eaton, 1992). The flung particles would react in an uncontrolled, locally fuel-lean environment (Wicker and Eaton, 1999). Although the relation between stoichiometry and particle dispersion is easy to understand, it remains to be exploited as a route toward improved burner design. Since one of the problems of Low NO_x burners (LNB) is the requirements of maintaining a near

burner fuel rich environment for the pyrolysis reactions to run their course, tailoring near-field vortices to obtain desired near-field particle concentration will definitely improve the capability of current LNBs.

Previous experimental studies have demonstrated that it is possible to control particle dispersion and subsequent stoichiometry through the manipulation of near-field large structures of the jet (Wicker and Eaton, 2001; Wark et al., 2000, Park et al., 1998). They have demonstrated that two phase non-reacting jets at loading representative of coal combustors, can be forced into ring vortices near the injector and with sub-harmonic forcing (active control), these vortices can be made to pair within prescribed distances from the injector. Since the rollup of these vortices are the primary mechanisms for air entrainment, controlling their positions and formation could be effectively used to effectively control the local stoichiometry. Active control might be too expensive or complex to implement in practice. However, passive control using asymmetric burner geometry such as elliptic nozzle may be a preferable alternative. Injection of longitudinal vortices in the jet shear layers from an elliptic coflow can control the dispersion of coal particles since longitudinal vortices are formed in the elliptic jet due to asymmetric self-induction of the vortex ring structures (Ho and Gutmark, 1987; Hussain and Hussain, 1989, Choudhuri et al., 2001). Non-axisymmetric nozzles are also characterized by increased fluid entrainment and better turbulent mixing as compared to axisymmetric jets. A non-axisymmetric jet provides better mixing due to a phenomenon known as axis switching that occurs further downstream of the nozzle exit. Increased fluid entrainment results in improved thrust augmentation. Turbulent mixing and entrainment are both desirable in combustion applications and can be obtained through a non-axisymmetric nozzle. The overall project goal is to investigate a passive control strategy of particle dispersion in coal-laden jets using asymmetric (elliptic) co flows. This particular project was aimed at characterizing the near-field behavior of single phase coaxial elliptic jets to understand more complex particle flow-field interactions in coal laden jets.

1.1 Objectives

The objective of this research is to numerically and experimentally investigate the near-field flow characteristics of a turbulent elliptical coaxial jet for developing passive particle dispersion control system for low NO_x burners. To design an elliptic coaxial flow nozzle, flows through two-dimensional and three-dimensional coflow nozzles are also calculated. The flow characteristics are also compared with a baseline condition (circular coflow). The specific objectives of this research project are:

1. To numerically compute annular flows through two-dimensional and three-dimensional circular and elliptic coaxial nozzles.
2. To numerically compute the near-field flow characteristics of circular and elliptic coaxial jets at different velocity ratios.

1.2 Project Overview

The project is divided into three phases to accomplish the research objectives. Each phase depends on the previous one. The phases begin with the annular design of the nozzle and ends with a coflow jet simulation.

- **Annular Design:** The annular design consists of a numerical analysis of a two-dimensional annulus of different outer wall geometries and lengths. The study is aimed at understanding the effects of nozzle contours and length on the annulus exit velocity profile. Flows through annuli of fifth, fourth, and third degree polynomial outer wall and fifth order polynomial inner wall were computed by solving the compressible Navier-Stokes equations over two dimensional

computational domains. The annular design aids in the design of the three-dimensional coaxial nozzles.

- Coaxial Nozzle Design: Three-dimensional numeric simulations of axisymmetric and non-axisymmetric co-axial jets are performed. These computations are performed to determine the exit annular velocity that will be used on the third phase of the project.
- Coflow Jet Analysis: In coflow jet analysis, a $k-\epsilon$ turbulent computational model of a coflow jet spreading into still ambient air is developed to understand the mixing characteristics of the coaxial nozzles. Both a circular and elliptical numerical model is developed. Experimental data is also gathered to validate the computational results. The coflow jet analysis provides information on the mixing mechanism produced by the axisymmetric and non-axisymmetric coaxial nozzles for the understanding of elliptical coflow on circular gas diffusion flames.

1.3 Organization of the Report

The approach utilized to fulfill the primary objective, and results obtained are discussed in seven chapters. Chapter 2 contains mixing characteristics of circular and elliptical jets, non-circular nozzles in combustion, annular flow in curved sections, and ends with circular and elliptical coflow. Chapter 3 contains the derivation of the governing equations, and the numerical methods used to discretization of governing equations. The turbulent equations employed in the numeric models are also explained within the chapter. The development of the Computer Aided Drawings (CAD) and the grid generation within the computational domain are presented in Chapter 4. Chapter 4 explains the boundary conditions and properties of the fluid used in the numeric simulation. The experimental methodology and the techniques of obtaining velocity data to validate the computed flow fields are in Chapter 5. The next chapter discusses the

numerical results. Chapter 6 is divided into two sections. Section 6-A discusses the results of the annuli and the coaxial nozzles. Section 6-B discusses the numeric and experimental results of the jets spreading in ambient air. Finally, the report is finalized with conclusions and recommendations.

Chapter 2

LITERATURE REVIEW

This chapter contains the necessary background relevant to this research. The chapter begins with an introduction to jet theory. This section is broken into two sections, non-reacting and reacting jets. The properties of laminar and turbulent diffusion flames are presented in the reacting jets section. Numerous literature is found on axisymmetric jets and its uses. The important contributions on axisymmetric jets are found in the section tilted non-reacting jets. Interested readers may refer to literature pertaining to axisymmetric jets for further information. The chapter then contains the mixing behavior of asymmetric jets and its uses in combustion applications. A brief section on asymmetric reacting jet and the formation of soot are also discussed. A background on annular flow is given with its application on coaxial jet design. The use of circular and elliptical coflow on diffusion flames concludes the chapter.

2.1 Jet Theory

The fluid expelled by a nozzle forms a free shear layer. Free shear layers are unaffected by walls and develop and spread in an open ambient fluid. Shear layers possess velocity gradients, created by some upstream mechanism, which try to smooth out by viscous diffusion in the presence of convective deceleration (White 1991). The plane free shear flow satisfies the flat plate equations. The continuity and momentum equations are as follows.

Continuity:
$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad [2-1]$$

X – Momentum:
$$u \frac{\partial u}{\partial x} + v \frac{\partial v}{\partial y} \approx \nu \frac{\partial^2 u}{\partial^2 x} \quad [2-2]$$

Equation [2-1] is formulated assuming the Reynolds number to be large, velocity of the jet is in the x direction, and the pressure gradient dP/dx is zero because there are no confining walls. The flow of a jet develops at the exit plane and maintains a uniform profile velocity distribution further downstream (White 1991).

An axisymmetric nozzle is a nozzle that is symmetrical about an axis. An example is a nozzle with a circular cross section. The fluid exiting the nozzle grows slowly, the radial change being much larger than axial change. Schlichting (1933) found a solution for a narrow laminar axisymmetric jet. The continuity and momentum equations that satisfy this flow are:

Continuity:
$$\frac{\partial u}{\partial x} + \frac{1}{r} \frac{\partial r}{\partial y} (rv) = 0 \quad [2-3]$$

X Momentum:
$$u \frac{\partial u}{\partial x} + v \frac{\partial v}{\partial r} = \frac{\nu}{r} \frac{\partial}{\partial r} \left(r \frac{\partial u}{\partial r} \right) \quad [2-4]$$

Schlichting (1933) reasoned that the jet thickness grew linearly and a similarity variable $\eta = r/x$ was defined. With this similarity variable, an axial velocity distribution equation was formulated. The velocity equation depends on the exit jet velocity, momentum flux, and a constant C .

Velocity Equation:
$$u = \frac{3J}{8\pi\mu x} \left(1 + \frac{C^2 \eta^2}{4} \right)^{-2} \quad [2-5]$$

$$\text{Momentum Flux:} \quad J = \frac{16\pi}{3} \rho C^2 v^2 \quad [2-6]$$

$$\text{Constant } C: \quad C = \sqrt{\frac{3J}{16\pi\rho v^2}} \quad [2-7]$$

Realistically, jets are turbulent and have more vigorous mixing with the surrounding fluid. Consequently, there is more entrainment. Fluid is entrained from the ambient into the jet, thus mass is increased downstream of the jet. A jet is divided into four zones as shown in *Figure 2-1*. The first zone is called the core, where the jet exits as fully developed turbulent flow with a top hat velocity distribution profile (*Figure 2-2*). In this region, the centerline velocity is equal to the exit velocity (White 1991). The velocity starts to decrease at a distance of one diameter from the exit in the transition zone (White 1991). The flow then maintains a distinctive Gaussian shape in the third zone. This occurs up to 20 diameters downstream of the jet (White 1991). The centerline velocity in the third zone can be calculated with the following equation:

$$\frac{U_c}{U_i} = K \frac{d_i}{x - x_p}, \quad [2-8]$$

where U_c is the centerline velocity along the axial direction and U_i is the outlet velocity of the jet. K is known as the velocity decay coefficient, d_i the exit diameter, x is the coordinate in the axial direction, and x_p is the distance measured from the exit of the jet to any point along the streamline. The termination zone is the last zone where the centerline velocity decreases rapidly.

Figure 2-1. Different zones of a spreading jet (Malmstrom et al. 1997).

Figure 2-2. Top hat velocity distribution at the jet's exit (White 1990).

2.1.1 Non-Reacting Jets

Research indicates that jet precision has proved to be beneficial in combustion applications (Ramesh 2000). These applications normally utilize circular nozzles. Extensive studies on circular jets have been performed in the past (Crow & Champagne

1971). A circular jet at low Reynolds number produces a thick boundary layer in which the exit profile resembles a Poiseuille pipe flow (Crow & Champagne 1971). Flow visualization experiments conducted by Crow and Champagne (1971) demonstrate the behavior of a jet with an increase in Reynolds number.

Figure 2-3 depicts the four different behaviors of a jet. The sketch is not associated with a specific Reynolds numbers. A whiplash motions is observed at very low Reynolds numbers (*a*). Crow and Champagne (1971) further state that the evolution from sinuous to pulsatile flow is complete at a Reynolds number of 10^3 . As the Reynolds number advances toward that value, the sinuous column coils into a corkscrew shape (*b*), then tightens and forms bulbous lobes rather like a crankshaft (*c*), and finally breaks into a train of axisymmetric puffs (*d*).

Figure 2-3. *Evolution of a jet's instability with advancing Reynolds number. Parts a) – d) span the Reynolds number from around 10^2 to 10^3 (Crow & Champagne 1971).*

2.1.2 Reacting Jets

In jet flames, the fuel flows along the flame axis and diffuses radially outward, while the oxidizer (air) diffuses radially inward. These types of flames are called diffusion flames. Diffusion flames are those in which the chemical reaction rate is very high compared to the rate of mixing, both molecular and turbulent diffusion. In diffusion flames, the fuel and oxidizer are initially unmixed. An ideal diffusion flame consists of an infinitesimally thin exothermic reaction zone, which separates the fuel stream from the oxidant stream. In this thin reaction zone, a high temperature reaction zone occurs in an annular region until the flame tip is reached (Turns 1996). The region where fuel and air meet in stoichiometric proportions is defined as the flame surface (Turns 1996). The products formed at the flame surface diffuse both radially inward and outward.

Diffusion flames are either homogeneous or heterogeneous. In homogeneous diffusion flames, the fuel as well as the oxidant is in gaseous state. Heterogeneous diffusion flames occur when a solid or liquid fuel burns in a gaseous oxidizing atmosphere. Apart from homogeneous and heterogeneous, diffusion flames are also classified as laminar or turbulent. The Reynolds number determines whether the flame is laminar or turbulent.

Laminar gas jet diffusion flames are fundamental to combustion and are researched intensively to understand the chemical kinetics and combustion characteristics (Savas and Gollahalli 1986). These flames involve complex interaction of thermo-physical processes and multi-step exothermic chemical reactions. Laminar diffusion flames are used in many residential gas appliances such as cooking ranges and ovens employ laminar diffusion flames (Turns 1996). Pollutant emission, flame structure, and flame geometry are the most important parameters in designing any system utilizing laminar diffusion flames.

Turbulent diffusion flames are mainly used in industrial applications. Visually, the flame has feathery or bushy edges. Turbulent diffusion flames are characterized by continuous fluctuation of velocity, which can lead to fluctuations in scalar such as density, temperature, and mixture composition (Warntz et al. 1996). When the fuel exits from a burner into surrounding air, the flame becomes turbulent with an increase in velocity. In the first few diameters from the burner, the flow is described as an axisymmetric shear layer. The central core fluid that exists on one side of the shear layer is constantly diluted by mixing with surrounding air. The flame length increases with velocity until the turbulent flow causes the air entrainment to increase at the same rate as the fuel input, with the result that the turbulent flame length is independent of jet velocity (Warntz et al. 1996).

2.2 Asymmetric Jets

An asymmetric jet is characterized by its vigorous turbulent mixing, axis switching, and mass entrainment. Injection or formation of longitudinal vortices can effectively change the pattern of large structures (Wicker and Eaton 1999). Longitudinal vortices are formed in an elliptical jet as a result of the radial and azimuthal instability present in the asymmetric jet exit. Commonly researched asymmetric jets include ellipses, rectangles, and triangles of aspect ratios below 5:1. For mixing process, a large mass entrainment, especially near the nozzle is desired (Ho & Gutmark 1987). Studies have found that improved mixing is achieved through an elliptical nozzle. Elliptical jets are superior to circular jets in fuel rich plume combustion process due to the entrainment of reactants (Schadow et. al 1987). Ho and Gutmark (1987 & 1985) reported that an elliptical jet with an aspect ratio of 2:1 entrains three to eight times more mass than an axisymmetric jet.

Sfroza et al. (1966) and Quinn (1989) report on three distinct regions of a non-axisymmetric (elliptical) jet. The first region, the potential core, is similar to that of a circular jet. In this region, the axial velocity is equal to the exit jet velocity. The next region is the characteristic decay region, where the axial velocity is dependent on the geometry of the jet exit. The third region, axisymmetric type decay, is the region where the flow approaches symmetry and velocity profiles in the minor and major axes are similar.

Elliptical nozzles are more complex than circular nozzles because of the two axes. Elliptical jets exhibit a phenomenon known as axis switching in which the major axis becomes the minor axis farther downstream from the nozzle. Axis switching has been documented by Zama (1996), Schadow et al. (1989), Hussain and Husain (1989), and Ho and Gutmark (1987). Axis switching tends to remove its initial instability by becoming symmetric downstream. The different growth rates of the major and minor axes and axis switching of elliptical jets lead to more entrainment of the surrounding air. *Figure 2-4* is an illustration of axis switching that occurs with an elliptical jet. Hussain & Husain (1989) describe axis switching as follows: as the major axis shrinks, it brings in surrounding air towards the jet centerline and at the same time, jet fluid is carried outwards by the expansion of the minor axis. The elliptical structures thus act as a pumping device to mix ambient and core fluid. This pumping action produces higher mixing in elliptical than in circular jets. Axis switching documented by Hussain and Husain (1989) occurred up to $100D_e$ ($D_e = 2(ab)^{1/2}$ for an elliptical nozzle exit).

Figure 2-4. Sketch of the development of axis switching (Hussain & Husain 1989).

This pumping action also explained by Ho and Gutmark (1987) occurs when the shear layer in the major axis plane spreads into the potential core while the shear layer in the minor axis plane spreads into the inactive surrounding. The momentum thickness in the minor axis plane becomes thicker than that in the major axis plane in a very short distance from the nozzle. As reported by Ho and Gutmark (1987), the growth rate in the minor axis plane is about 20% more than that in the major axis plane. Both values are much higher than the growth rate of a circular nozzle. The large entrainment of the elliptical jet was found to be mainly produced in the portion near the minor axis, whereas the region near the major axis, the entrainment is approximately the same as that of the circular jet (Ho & Gutmark 1987).

Numerical simulations on non-circular jets were performed by Miller, Madnia, and Givi (1994). The ellipse in this study had a 2:1 aspect ratio. Results show a contraction of flow in the major axis plane and spreading in the minor axis plane. Axis switching was also observed in the numeric simulation. The location of axis switching was at $x/D_e \approx 3$. The experimental results obtained by Hussain and Husain (1989) indicate an axis switching at a $x/D_e \approx 2.8$ and Ho and Gutmark (1987) determined axis switching at a $x/D_e \approx 2.4$. Numerical result from Miller, Madnia, and Givi (1994) are in good agreement with the experimental results performed by others.

2.2.1 Asymmetric Reacting Jets

Work on non-circular jets in combustion is limited. Schadow et al. (1984) determined that combustion efficiency in elliptical jets is approximately 10% higher than circular jets. This results from the increase of entrainment provided by the elliptical jet. Experiments on noncircular burners performed by Gollahalli (1997) indicate that

elliptical burners consume more fuel than circular burners. The rapid depletion of fuel was caused by the greater dilution due to the higher entrainment in the elliptical nozzle flame (Gollahalli 1997). Elliptical flames produce slightly less nitric oxide concentrations and less soot due to the large availability of oxygen (Gollahalli 1997). Nitric oxide (NO) and nitrogen dioxide (NO₂) are the major nitrogen pollutants. Photochemical smog, global warming, and acid rain are caused by nitrogen oxides (NO_x).

Soot is the intermediate product of combustion that gives the diffusion flame its luminosity. Soot is undesirable because it causes radiant heat loss, and makes the flame smoky. Soot in diffusion flames is formed over a range of $1300\text{K} < T < 1600\text{K}$ (Turns 1996 cite). The formation of soot occurs in a four step process know as:

- Formation of precursor species
- Particle inception
- Surface growth and particle agglomeration
- Particle oxidation

In the first step of soot formation, chemical kinetics plays an important role in which the PAH (polycyclic aromatic hydrocarbons), the intermediate species between the original fuel molecules and the primary soot particle are formed. In the second step, large molecules are transformed to particles ranging from 3,000 to 10,000 atomic mass units. Following particle inception, the particles undergo surface growth and agglomeration due to its exposure in a stream of species generation from fuel pyrolization. The final step consists of the soot particles oxidizing in the flame tip, if the flame is issued from a jet (Turns 1996).

2.3 Annular Flow

Annular flow occurs in many applications, mainly in heat exchangers, and is used to connect between the high and low-pressure stages of a turbine. Annular jets have applications in sprays and plastic blow molding processes. Limited data is found in explaining how curvature and length affects annular flow. Ramos (2001) formulated leading-order fluid dynamic equations of isothermal, axisymmetric, Newtonian, annular liquid jets at low Reynolds numbers. Studies have concentrated on flows in curved pipes. Curved sections in pipes occur in piping systems, and it is important to know the pressure drop in the developing and fully developed parts of the flow if one is to predict the pumping power needed to overcome curvature induced pressure losses (Berger et al. 1983). The principles of flow in curved channels also apply to arteries. The aorta, a blood vessel curved three-dimensionally through 180° has received much attention in recent years (Jain and Jayarman 1990).

Brithchford et al. (1994) conducted measurements and computations on annular S-shaped ducts. The results show that upon entering the S-shaped portion, the flow near the inner wall accelerates at first, and then decelerates due to the thickening of the inner wall boundary layer. A symmetrical velocity profile was then achieved as the flow exited the duct. Jain and Jayaraman (1990) studied laminar flow in curved pipes with elliptical cross section, and developed a mathematical formulation based on a Newtonian and incompressible fluid with a constant density and viscosity. Their results indicate a velocity deceleration towards the outer bend and an acceleration near the inner bend.

2.3.1 Coaxial Jets

The importance of annular flow is applied in the design of coaxial jets. A challenging task lies in producing a uniform exit annular velocity when designing an outer nozzle. Coaxial jets (*Figure 2-5*), in which the central jet is surrounded by an annular jet, are researched extensively due to their practical application to the areas of propulsion and combustion in which fuel and air are to be mixed. The mixing of two coaxially discharged fluids occurs due to the interaction between the shear layers of the fluid. The mixing rate increases due to the interaction of the two fluids, thus mass is increased. In combustion applications, a fuel stream is mixed with a coaxial air stream. This is performed in order to increase the oxygen content of the fuel by enhancing the mixing rates of the fuel and air.

Figure 2-5. *A front and back view of axisymmetric coaxial jet.*

Experimental results on coaxial turbulent jets obtained by Champagne and Wygnanski (1971) describe the development of the jet. The initial flow close to the nozzle exit consists of two potential cores separated by an annular mixing region and another mixing region between the outer jet and the ambient air (*Figure 2-6*). The width of each core decreases approximately linearly with downstream distance and the cores terminate when

the annular mixing region joins. Downstream, the flow becomes fully developed and self-preserving and the velocity at any point depends only on the position of the point and the total jet momentum.

Figure 2-6. Flow development in a coaxial jet (Champagne and Wygnanski 1971).

Further studies performed by Ko and Au (1985) on mean velocity ratios (inner to outer exit nozzle velocity) ranging from $0.15 \leq U_i/U_o \leq 1$ reveal three zones within the coaxial jet: initial merging zone, intermediate merging zone, fully merge zone (Figure 2-7). The first zone is called the initial merging zone, which represents the region from the nozzle exit to the end of the outer potential core. The intermediate zone extends from the end of the outer core to the reattachment point in which the inverted velocity profile in the inner jet region vanishes (Tang & Ko 1994). This is the region with the most momentum transfer between the two streams of fluid. The fully merged zone begins at the reattachment point, which is the point where the maximum in the mean velocity profile returns to the jet centerline.

Figure 2-7. *The three zones of a coaxial jet (Tang & Ko 1994).*

The evolution of a coaxial jet and the mixing characteristics are influenced by the velocity ratios ($m = U_o/U_i$). Au and Ko (1985 and 1987) conducted a two-part investigation on the effect of mean velocity ratios (m^{-1}) ranging from 0.15 to 1.0. Tang and Ko (1993) and Ko and Lam (1989) investigated mean velocity ratios (m^{-1}) of 0.3 and 0.5 respectively. Most recently, Villiermaux and Rehab (2000) studied coaxial jets with a large outer (annular) to inner velocity ratio to see its effect on mixing, and Buresti, Petagna, and Talamelli (1998) conducted LDA and hot-wire measurements for $m^{-1} = 0.3$ and $m^{-1} = 0.67$.

These investigations attempted to explain the near field behavior of coaxial jet and annular jets in terms of the vortex structures in the two mixing layers using velocity and pressure measurements. The authors conclude that a mean velocity ratio of $m^{-1} = 0.3$ produces two core of comparable size with having highest turbulent intensity, and higher strength of the inner shear layer which favors a more rapid and intense mixing of flows. For $U_i/U_o = 0.3$ and $U_i/U_o = 0.5$, a train of vortex rings develop in the outer and inner

shear layers. A ratio of $m = 1$ results in the merging of the two cores into one due to the disappearance of the upstream boundary layer (Champagne and Wygnanski 1971).

There are two signs of vorticity merging in the shear layer due to the viscous boundary layers on both sides of the inner nozzle (Wicker and Eaton 1993). The vortices produced from the annular jet merge with those of the inner jet. A second shear layer develops from the interaction of the annular flow with the ambient. Within the outer mixing region of the coaxial jet the initial vortex sheets rolls up forming vortices with an outwards sense of rotation. These vortices amalgamate into bigger ones slightly further downstream, forming the outer structures, and convect downstream into the intermediate merging and fully merged zones. These outer structures, which are found to be similar to the coherent structure existing in single jets, are not significantly affected by the change of mean velocity ratios (Ko and Au 1985). The core of the outer flow is also independent of the velocity ratio and is equal to approximately 8 times the width of the annular nozzle (Champagne and Wygnanski 1971).

An inner shear layer develops between the core and annular flow. The inner structure (inner vortices) characteristics depended on the jet velocity ratio, with limiting cases of an annular jet for large m and a single jet for small m (Au and Ko 1987, Lam and Ko 1986). The length of the inner core is also dependent on the mean velocity ratio (Champagne and Wygnanski 1971). Understanding vortices merging and mean velocity ratios from coaxial jet, which augment mixing, can enhance combustion applications.

2.4 Coflow

Coflow in combustion is a method whereby air from an annular jet is supplied to the flame that is ejected from a central jet (Figure 2-8). This is performed in order to

increase the oxygen content in the flame by enhancing the mixing rates of the fuel and air. Air is entrained into the fuel core through molecular diffusion and turbulent mixing. Thus, flame structure is dependent on flow characteristics. Controlling the mixing rate between air and fuel can reduce pollutants such as nitric oxides, sulfur oxides, carbon monoxide, and carbon dioxide.

Recent contributions on the understanding of coflow have been performed experimentally and computationally. Bennet et al. (2001) performed computational and experimental studies on axisymmetric coflow on partially premixed flames. Huang, Yang and Lee (1997) researched flame and flow characteristics in which air was supplied by the central jet and a fuel was ejected from the annular jet. Choudhuri et al. (2002) studied the effects of elliptical coflow on turbulent gas jet diffusion flames.

Figure 2-8. Schematic of coflow on combustion.

Although, the study observed a higher NO_x production by an elliptic coflow in diffusion flames (homogeneous gas phase combustion), however, the higher entrainment and highly asymmetric near-burner flow-field might yield a positive result for heterogeneous combustion in coal-laden flows. The study observed that flames with elliptical coflow showed steeper centerline velocity decay compared to the circular coflow flames. This feature of the flow-field was attributed to the increased entrainment through the injection of longitudinal vortices inside the inner shear layer by the elliptic outer nozzle. Although, these characteristics caused high nitric concentration resulting from high temperature combustion (due to higher availability of oxygen in reaction zone) in diffusion flames, the same feature can be effectively used to tailor the near-field sequence of stoichiometry of a coal-laden jet to achieve aerodynamic NO_x abatement. Since maintaining aggregate particle cloud in a near-burner region lowers NO_x production, the contra-rotating pairs of longitudinal vortices shed by the corner of the elliptic jet could be effectively used to control the local stoichiometry. The rollup of these vortices are the primary mechanisms for air entrainment, hence controlling their positions and formation will allow tailoring the local stoichiometry of the coal suspension to reduce NO_x formation.

2.5 Low NO_x Burner Technology

The emission of oxides of nitrogen (NO_x) pollutants has wide range of effects on the environment which include acid rain, reduction of atmospheric visibility, production of tropospheric ozone, and in the case of N_2O , depletion of ozone layer. Nitric oxide (NO) is formed in fuel lean flames by the combination reaction of O atom and molecular nitrogen. In fuel-rich flames it forms through the reaction of N_2 and hydrocarbon radicals and by the pyrolysis and oxidation of heterocyclic nitrogen compounds in coal. *Figure 2-9* shows the chemical pathways of nitrogen compound interconversion in fuel lean and fuel rich flame. It is believed that the necessary NO_x reduction mechanism can be obtained

through a combination of in-furnace combustion modification, low NO_x burner, reburn, etc. without the need of expensive flue gas treatment process. The development of chemical reaction pathways of nitrogen oxides formation and destruction in flames helps the formulation of strategies for primary measures of NO_x emission reduction through combustion modifications. The strategies can be summarized as follows (Beer, 2000):

Figure 2-9. Chemical Pathways of NO_x Formation and Destruction (Beer, 2000)

1. Reduction of peak flame temperature by heat extraction or flue gas recirculation.
2. Dilution of reactants
3. Staging of combustion air to produce fuel-rich/fuel-lean sequencing to arrange favorable conversion of fuel bound nitrogen to N_2
4. Staging of fuel to arrange NO reburning

Through the last two decades the use of NO_x reduction technique through combustion process modification has been increased significantly. Currently more than 188 GW of electric power generating capacity used internationally has been fitted by these ‘primary combustion modification measures’. Combustion modified low NO_x represents the most cost effective method of achieving reduced NO_x emission from new plant boilers, and

also existing boilers by retrofit. Additionally low NO_x burner (LNB) systems have the benefit of being easy to maintained one installed and do not require the handling of toxic chemicals, such as ammonia, associated with flue gas NO_x treatment techniques, furthermore, it should not result in an increase in unburned carbon. *Figure 2-10* shows the schematic of Low NO_x burner concepts. *Figure 2-11* shows the system schematics of a low NO_x burners recently developed by B&W.

One of the problems of LNBs is the requirement of maintaining a fuel-rich zone close to the burner for the pyrolysis reactions to occur, followed by a fuel-lean region (Strategy 3) to complete the combustion process. The large-scale fluid dynamic structures present in the near burner flow can fling the particles outward or even completely outside the outer shear layer. The flung particles would react in an uncontrolled, locally fuel-lean environment (Wicker and Eaton, 1999). Similarly, nearly axisymmetric vortex rings can also produce instantaneous regions of high particle concentration. Although advanced active combustion control techniques such as acoustic forcing or flow modulation can be used to sequence the desired stoichiometry, elliptic coaxial nozzle bases passive control may be a cost effective solution for this problem. The injection of longitudinal vortices in the jet shear layers from an elliptic coflow can be used to control and organize the near burner vorticial structures in order to reduce preferential concentration of the coal

Figure 2-10. Low NO_x Burner Concept

particles. Although in theory it is possible to control particle stoichiometry through elliptic coaxial flows, it remains to be exploited as a route toward improved burner design.

Figure 2-11. Low NO_x DRB-4ZTM Burner (Credit B & W)

Chapter 3

Numerical Methods

The numerical algorithms employed in computational fluid dynamics are based on the governing equations of fluid dynamics - the continuity, momentum, and energy equation. These equations are used to generate the numerical codes. The numerical codes replace these 2nd order non-linear equations into discretized algebraic forms. An understanding of the governing equations is necessary to understand the discretization techniques.

3.1 Governing Equations

The governing equations are based on fundamental physical principles: mass is conserved, Newton's second law, and energy is conserved. The mathematical notations used to derive the governing equation are presented below. The velocity vector in the x , y , and z components is represented by a capital $\mathbf{V}$.

$$\mathbf{V} = u\mathbf{i} + v\mathbf{j} + w\mathbf{k} \quad [3-1]$$

The vector operator ∇ is defined as

$$\nabla = \frac{\partial}{\partial x} + \frac{\partial}{\partial y} + \frac{\partial}{\partial z} \quad [3-2]$$

The divergence of the velocity vector field is written as $\nabla \cdot \mathbf{V}$ or $\text{div}\mathbf{V}$ where,

$$\nabla \cdot \mathbf{V} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \quad [3-3]$$

The Laplacian, $\nabla \cdot \nabla \mathbf{V} = \nabla^2 \mathbf{V}$, is defined as the divergence of the gradient and can be written as *grad*($\mathbf{V}$). Expanding the Laplacian term results in the following equation

$$\nabla^2 \mathbf{V} = \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \quad [3-4]$$

Another term used in the derivation of the governing equations is the substantial derivative. The substantial derivative, D/Dt , is equal to the local ($\partial/\partial t$) and convective derivative ($\mathbf{V} \cdot \nabla$). The substantial derivative represents the time rate of change, which accompanies the moving fluid element. The local derivative is the time rate of change at a specific point. Finally, the convective derivative is the time rate of change of the fluid element due to its movement from one location to another where the flow properties are different. The substantial derivative is written as

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + \mathbf{V} \cdot \nabla \quad [3-5]$$

Expanding equation [3-5] yields

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + u \frac{\partial}{\partial x} + v \frac{\partial}{\partial y} + w \frac{\partial}{\partial z} \quad [3-6]$$

3.1.1 Continuity Equation

The continuity equation states that mass within the control volume can change with time if mass is added or removed. The mass flow rate is defined as $\dot{m} = \rho A \vec{v}$ for two-

dimensional flow. When there is no change in mass within the control volume, the mass entering the control volume is equal to the mass exiting the control volume.

Consider a three-dimensional fluid element as shown in *Figure 3-1* with a volume of $dxdydz$.

Figure 3-1. Fluid element of volume $dxdydz$ with mass fluxes through various faces (Anderson 1995).

The total mass of the fluid element is $m = \rho(dxdydz)$. The continuity equation can be expressed as

$$\left(\begin{array}{l} \text{Rate of decrease of mass} \\ \text{in a control volume} \end{array} \right) = \left(\begin{array}{l} \text{Net rate of mass flow out} \\ \text{of control volume} \end{array} \right)$$

In mathematical terms, the above statement is expressed as:

$$\text{Time rate of decrease of mass} = -\frac{\partial \rho}{\partial t}(dxdydz) \quad [3-7]$$

$$\text{Net mass flow} = \left[\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} \right] dxdydz \quad [3-8]$$

Equating [3-7] and [3-8] yields

$$-\frac{\partial \rho}{\partial t}(dxdydz) = \left[\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} \right] dxdydz$$

$$\frac{\partial \rho}{\partial t} + \left[\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} \right] = 0 \quad [3-9]$$

The terms in brackets can be written as $\nabla \cdot (\rho \mathbf{V})$ or the $div(\rho \mathbf{V})$. Thus, equations [3-9] simplifies to

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0 \quad [3-10]$$

For incompressible flows in which density is constant, equation [3-10] reduces to

$$div \mathbf{V} = 0$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad [3-11]$$

3.1.2 Momentum Equation

The momentum equation is based on Newton's second law, which state that the sum of the forces is equal to the rate of change of momentum in a given direction as is expressed as

$$\sum F_x = ma_x \quad [3-12]$$

The forces acting on a fluid element are classified as surface forces and body forces. Surface forces act directly on the surface of the fluid element. These forces include shear and normal stresses acting on the surface of the element known as viscous forces.

Pressure forces are another type of body forces that act on the faces of the fluid element caused by a pressure distribution. Body forces act directly on the volumetric mass of the fluid element and include gravitational and electromagnetic forces. Consider the fluid element in *Figure 3-2* in which velocity is moving in the x-direction. The pressure forces, normal and shear stress are applied at the appropriate face.

Figure 3-2. Forces in the x-direction acting on the faces of fluid element (Anderson 1995).

The summation of forces is as follows.

$$\sum F_x = F_{surface} + F_{body}$$

$$\sum F_x = \left[\left(-\frac{\partial P}{\partial x} \right) dx dy dz + \left(\frac{\partial \sigma_x}{\partial x} \right) dx dy dz + \left(\frac{\partial \tau_{yx}}{\partial y} \right) dx dy dz + \left(\frac{\partial \tau_{zx}}{\partial z} \right) dx dy dz \right] + \rho f_x dx dy dz \quad [3-13]$$

Recalling that mass is equal to $m = \rho(dx dy dz)$, and that the rate of change of velocity (acceleration) is written as

$$a_x = \frac{Du}{Dt} \quad [3-14]$$

Substituting equations [3-13], [3-14], and the mass term into [3-12] yields the momentum equation in the x-direction.

$$\left(\begin{array}{c} \text{Rate of change of momentum} \\ \text{of a fluid particle} \end{array} \right) = \left(\begin{array}{c} \text{Sum of forces acting} \\ \text{on the fluid particle} \end{array} \right)$$

$$\rho \frac{Du}{Dt} = -\frac{\partial P}{\partial x} + \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x \quad [3-15]$$

Similarly, the momentum equations in the y and z components are

$$\rho \frac{Dv}{Dt} = -\frac{\partial P}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y \quad [3-16]$$

$$\rho \frac{Dw}{Dt} = -\frac{\partial P}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + \rho f_z \quad [3-17]$$

Expanding the substantial derivative in equation [3-15] yields

$$\rho \frac{Du}{Dt} = \frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho u \mathbf{V}) \quad [3-18]$$

Expanding the stress components as shown below

$$\begin{aligned} \tau_{yx} = \tau_{xy} &= \mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) & \sigma_x &= 2\mu \frac{\partial u}{\partial x} \\ \tau_{zx} = \tau_{xz} &= \mu \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) & \sigma_y &= 2\mu \frac{\partial v}{\partial y} \end{aligned}$$

$$\tau_{zy} = \tau_{yz} = \mu \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) \qquad \sigma_z = 2\mu \frac{\partial w}{\partial z}$$

and substituting the respective stress components into equations [3-15], [3-16] and [3-17], with its corresponding coordinates yields the following equations known as the Navier-Stokes equations for Newtonian fluid.

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + z \frac{\partial u}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \nu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) + f_x \quad [3-19]$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + z \frac{\partial v}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \nu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + f_y \quad [3-20]$$

$$\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + z \frac{\partial w}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \nu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + f_z \quad [3-21]$$

3.1.3 Energy Equation

The energy equation which derives from the first law of thermodynamics states the rate of change in energy is equal the rate of heat added to the fluid element plus the rate of net work done on the element. The mathematical relation is $dE = dq + dW$.

The same process of deriving the continuity and momentum equations is used to derive the energy equation. Starting from a fluid element, the components that do work on the surface of the element, and the heat flux that enters and leaves the faces of the element

are derived. Combining the heat flux and work components yields the energy equation in terms of flow variables.

$$\rho \frac{De}{Dt} = \rho \dot{q} - \text{div}(P\mathbf{V}) + \text{div}(k \text{ grad } T) + \lambda(\text{div}\mathbf{V})^2 + \mu \left[2\left(\frac{\partial u}{\partial x}\right)^2 + 2\left(\frac{\partial v}{\partial y}\right)^2 + 2\left(\frac{\partial w}{\partial z}\right)^2 + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)^2 \right] \quad [3-22]$$

In the above equation, e is the internal energy and is equal the kinetic and potential energy of the fluid element (equation [3-23]).

$$e = \mathbf{V}^2/2 + gz \quad [3-23]$$

To summarize, the governing equations that were derived are nonlinear second order partial differential equations. The stresses in the governing equations are functions of velocity, and the Navier-Stokes equations were derived from the momentum equations. As of now, no analytical solution exists for these types of equations. There are a total number of six variable ρ , P , u , v , w , e , and a total of five equations, which are the continuity equation, three momentum equations corresponding to the x , y , z components, and the energy equation. There are more variables than equations needed to solve the governing equations. To correct this problem, the equation of state adds two additional equations to solve for the unknowns. For a perfect gas, the following is true

$$P = \rho RT \quad [3-24]$$

and the internal energy is equal to the product of the specific heat of the fluid and its change in temperature,

$$e = c_v dT \quad [3-25]$$

Terms can be removed from the governing equations by making necessary assumptions and applying boundary conditions. The equation of state and the governing equations

provide velocity distribution functions that allow for a mathematical and visual representation of flow.

3.2 Turbulent Modeling

Most realistic flows are turbulent which result from the instability of laminar flows. Turbulent flow can be defined as random, three-dimensional flow, having a large Reynolds number. Turbulent flows are also dissipative. Viscous shear stresses perform deformation work, which increases the internal energy of the fluid at the expense of kinetic energy of turbulence (Tennekes and Lumley 1972). Due to its random nature, turbulent flow is described by its time-averaged properties (i.e. velocities, pressures, stresses). The Favre Averaged Navier Stokes Equations (FANS) is used to take into account the mean and the time varying fluctuations of the flow property. This averaging technique decomposes a flow quantity to its mean and fluctuating value.

$$\theta = \tilde{\theta} + \theta'' \quad \text{where } \tilde{\theta} = \frac{\overline{\rho\theta}}{\bar{\rho}} \quad [3-26]$$

Applying the Favre averaging procedure to continuity equation, the Favre-averaged continuity equation can be obtained and written as

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{u}_j) = 0 \quad [3-27]$$

Similarly, the Favre-averaged Navier Stokes Equations are

$$\frac{\partial}{\partial t}(\bar{\rho} \tilde{u}_i) + \frac{\partial}{\partial x_j}(\bar{\rho} \tilde{u}_i \tilde{u}_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\bar{\mu} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \tilde{u}_m}{\partial x_m} \delta_{ij} \right) \right] + \frac{\partial}{\partial x_j} (-\rho \widetilde{u_i' u_j'}) + \bar{\rho} f_i \quad [3-28]$$

In the present computation, the Boussinesq eddy viscosity concept was used in which the Reynolds stress $-\rho \widetilde{u_i' u_j'}$ is treated as a linear function of the mean strain rate

$$-\rho \widetilde{u_i' u_j'} = \mu_t \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \tilde{u}_m}{\partial x_m} \delta_{ij} \right) - \frac{2}{3} \bar{\rho} k \delta_{ij} \quad [3-29]$$

where μ_t is the turbulent eddy viscosity and k is one half the trace of the Reynolds stress tensor.

$$k = \frac{1}{2} (\widetilde{u_k' u_k'}) \quad [3-30]$$

By substituting the Reynolds stress term in the Navier Stokes equation, the following equation is obtained.

$$\begin{aligned} \frac{\partial}{\partial t}(\bar{\rho} \tilde{u}_i) + \frac{\partial}{\partial x_j}(\bar{\rho} \tilde{u}_i \tilde{u}_j) = & -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\bar{\mu} + \mu_t) \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \tilde{u}_m}{\partial x_m} \delta_{ij} \right) \right] \\ & - \frac{2}{3} \frac{\partial}{\partial x_i} (\bar{\rho} k) + \bar{\rho} f_i \end{aligned} \quad [3-31]$$

Similarly, when the averaging technique is applied to the static enthalpy equation, the averaged equation also contains a fluctuating enthalpy and velocity term, which also requires modeling through Boussinesq concept.

$$-\bar{\rho} \widetilde{u' h'} = \Gamma_t \frac{\partial \tilde{h}}{\partial x_j} \quad [3-32]$$

The term Γ_t is the turbulent or eddy diffusivity and is related to the turbulent viscosity through the turbulent Prandtl number σ_t

$$\Gamma_t = \frac{\mu_t}{\sigma_t} \quad [3-33]$$

The mean enthalpy equation is derived by adding the mean kinetic energy equation to the mean static enthalpy equation.

$$\begin{aligned} \frac{\partial \bar{\rho} \tilde{h}}{\partial t} + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{u}_j \tilde{h}) = \frac{\partial}{\partial x_j} \left\{ (\Gamma + \Gamma_t) \frac{\partial \tilde{h}}{\partial x_j} \right\} + \frac{\partial \bar{p}}{\partial t} + u_j \frac{\partial \bar{p}}{\partial x_j} \\ + (\bar{\mu} + \mu_t) \frac{\partial \tilde{u}_i}{\partial x_j} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \tilde{u}_k}{\partial x_k} \delta_{ij} \right) \end{aligned} \quad [3-34]$$

3.2.1 Turbulence Model $k-\varepsilon$

In the computational methodology applied in the present investigation, the effect of turbulence is accounted for via turbulent or eddy viscosity. Turbulent viscosity is not a property of a fluid but rather a characteristic of the flow. Following the kinetic theory of gases, the turbulent viscosity is modeled as the product of a velocity scale $\mathcal{G}$ and a length scale l ,

$$\mu_t = C \bar{\rho} \mathcal{G} l \quad [3-35]$$

where C is a constant of proportionality. Various turbulence models differ in the way $\mathcal{G}$ and l are estimated. In the present study, standard $k-\varepsilon$ model based on Launder and

Spalding (1974) was used. The k - ε model is considered one of the most validated and well established turbulent models. An advantage is its simplicity for which only initial and or boundary conditions need to be supplied (Versteeg & Malalasekera 1995). The k - ε turbulent model is based on the time-average Reynolds equations. This model employs two partial differential equations to estimate the velocity and length scales and hence commonly known as the two-equation model. These equations are the k and ε equations, which govern the transport of the turbulent kinetic energy and its dissipation rate respectively. The modeled equations for k and ε are

$$\frac{\partial(\overline{\rho k})}{\partial t} + \frac{\partial}{\partial x_j}(\overline{\rho u_j k}) = \overline{\rho P} - \overline{\rho \varepsilon} + \frac{\partial}{\partial x_j} \left[\frac{\overline{\mu} + \mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] \quad [3-36]$$

$$\frac{\partial(\overline{\rho \varepsilon})}{\partial t} + \frac{\partial}{\partial x_j}(\overline{\rho u_j \varepsilon}) = C_{\varepsilon_1} \frac{\overline{\rho P \varepsilon}}{k} - C_{\varepsilon_2} \frac{\overline{\rho \varepsilon^2}}{k} + \frac{\partial}{\partial x_j} \left[\frac{\overline{\mu} + \mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right] \quad [3-37]$$

with the production term P defined as

$$P = \overline{\rho \mu_t} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \tilde{u}_m}{\partial x_m} \delta_{ij} \right) \frac{\partial \tilde{u}_i}{\partial x_j} - \frac{2}{3} k \frac{\partial \tilde{u}_m}{\partial x_m} \quad [3-38]$$

Equations [3-36] states that the rate of change of turbulent kinetic energy plus the transport of k by convection is equal to rate of production of k minus the rate of destruction of k plus the transport of k due to diffusion. Similarly, equation [3-37] states the same but with regards to ε . Equation [3-37] assumes that its production and destruction term are proportional to the production and destruction terms of the k equation. This insures that the dissipation rate increases rapidly if k increases rapidly and decreases sufficiently fast to avoid (non physical) negative values of turbulent kinetic energy if k decreases (Versteeg & Malalasekera 1995).

The square root of k is taken to be the velocity scale ($\mathcal{G} = \sqrt{k}$) while the length scale is obtained from

$$l = \frac{C_u^{3/4} k^{3/2}}{\varepsilon} \quad [3-39]$$

The expression for turbulent viscosity is

$$\mu_t = \frac{\bar{\rho} C_\mu k^2}{\varepsilon} \quad [3-40]$$

The five constants used in the model are (Launder and Spalding 1974)

$$C_\mu = 0.09; \quad C_{\varepsilon_1} = 1.44; \quad C_{\varepsilon_2} = 1.92; \quad \sigma_k = 1.0; \quad \sigma_\varepsilon = 1.3$$

Approximations for k and ε used for inlet conditions are

$$k = \frac{3}{2} (U_{ref} T_i)^2 \quad [3-41]$$

and

$$\varepsilon = \frac{C_u^{3/4} k^{3/2}}{l} \quad [3-42]$$

where U_{ref} is the mean velocity in all three components, T_i is the turbulent intensity, and $l = 0.07L$ with L as the characteristic length.

3.3 Discretization Method

All governing equations, except the continuity equation can be expressed as a general combination of transient, convection, diffusion, and source terms.

$$\frac{\partial}{\partial t}(\rho\phi) + \text{div}(\rho\phi\mathbf{V}) = \text{div}(\Gamma \text{grad}\phi) + S_\phi \quad [3-43]$$

Transient + Convective = Diffusion + Source

In equation [3-43], ϕ is the dependent variable and may represent velocity components, enthalpy, or any scalar flow variable. S_ϕ is the source term, which represents the mechanisms for the generation or destruction of ϕ , and Γ is the diffusion coefficient. In the present computation, a finite volume approach, in which the governing equations are numerically integrated over the control volume is used. In this method, the finite difference equations (FDE) are formulated by evaluating and integrating fluxes across the faces of the control volume (*Figure 3-3*) in order to satisfy the conservation equation.

Figure 3-3. Control volume for a two-dimensional grid.

Integrating equation [3-43] over the control volume results in

$$\int_V \frac{\partial}{\partial t}(\rho\phi) dV + \int_V \text{div}(\rho\phi\mathbf{V}) dV = \int_V \text{div}(\Gamma \text{grad}\phi) dV + \int_V S_\phi dV \quad [3-44]$$

The next step in converting the general transport equation to a set of algebraic equations is to discretize each term in equation [3-44]. Discretizing the transient term yields

$$\int_V \frac{\partial}{\partial t} (\rho\phi) dV = \frac{\rho\phi V - \rho^o \phi^o V^o}{\Delta t} \quad [3-45]$$

where the superscript “o” denotes an older time, while no superscript represents the current time.

3.3.1 Convective Discretization

The convective term is discretized as follows,

$$\int_V \text{div}(\rho\phi\mathbf{V}) dV = \sum m_f \phi_f \quad [3-46]$$

where $m_f = \rho u$, is the mass flux across the face ($\mathbf{n}$, $\mathbf{s}$, $\mathbf{w}$, $\mathbf{e}$; refer to *Figure 3-3*) per unit area. The variable ϕ is evaluated at the center of the control volume and different numerical schemes are used to interpolate the value of ϕ at the faces of the control volume. The first order upwind scheme was the numerical scheme used. The upwind scheme utilizes consistent expressions to calculate fluxes through cell faces. A first order upwind scheme has a first order accuracy on the basis of the Taylor series, and is one of the most stable schemes (Versteeg & Malalasekera 1995). This scheme is widely used due to its simplicity and is effective when the flow is aligned with the grid lines, as is the case with the current computations.

3.3.1.1 Upwind Scheme

The upwind scheme takes into account the direction of flow when determining the value of the faces of the control volume. The convected value of ϕ is taken to be equal to the values of the upstream node (Versteeg & Malalasekera1995). *Figure 3-4* shows the convected values of ϕ at each face. In this figure, **P** is the center of the control volume, **E** and **W** are the neighboring nodes, and **e** and **w** are the control volume faces. If the flow is in the positive direction as seen in *Figure 3-4 a*), the values of the control volume faces become

$$\phi_w = \phi_W \quad \text{and} \quad \phi_e = \phi_P$$

Likewise, if the flow is negative (*Figure 3-4 b*)) then the values of the control faces become

$$\phi_w = \phi_P \quad \text{and} \quad \phi_e = \phi_E$$

Figure 3-4. Discretized domain used in the upwind scheme with flow entering the control volumes a) positive flow b) negative flow.

3.3.2 Diffusion Discretization

Consider a one-dimensional steady state diffusion problem. *Figure 3-5* shows the discretized domain, where P is the center node of the control volume with E and W being the east and west nodal points respectively. The east and west faces of the control volume are represented as e and w respectively. The distance between W and P is denoted as δ_W , and the distance between E and P is represented as δ_E .

Figure 3-5. Discretized domain with control volume.

The discretized diffusion equation for this one-dimensional example with no source term is

$$\int_V \frac{d}{dx} \left(\Gamma \frac{d\phi}{dx} \right) = \left(\Gamma A \frac{d\phi}{dx} \right)_e - \left(\Gamma A \frac{d\phi}{dx} \right)_w$$

$$\left(\Gamma A \frac{d\phi}{dx} \right)_e - \left(\Gamma A \frac{d\phi}{dx} \right)_w = 0 \quad [3-47]$$

The west and east fluxes are then discretized to the following

$$\left(\Gamma A \frac{d\phi}{dx} \right)_e = \Gamma_e A_e \left(\frac{\phi_E - \phi_P}{\delta_E} \right) \quad [3-48]$$

$$\left(\Gamma A \frac{d\phi}{dx} \right)_w = \Gamma_w A_w \left(\frac{\phi_P - \phi_W}{\delta_W} \right) \quad [3-49]$$

Substituting the fluxes into equation [3-48] and simplifying yields the following

$$a_P \phi_P = a_W \phi_W + a_E \phi_E \quad [3-50]$$

If the flow contains a source term, the term is added to the left hand side of the equation. For two- and three-dimensional diffusion problems the general discretized equation is

$$a_P \phi_P = \sum a_{nb} \phi_{nb} + S_u \quad [3-51]$$

where a_{nb} are the neighboring coefficients (east, west, north, south ...).

3.3.3 Source Term Linearization

The source term, which is a function of ϕ , is linearized as (Patankar, 1980)

$$S_\phi = S_\phi^P \phi + S^U \quad [3-52]$$

such that S^P is negative. S^P and S^U can be functions of ϕ . These coefficients are evaluated using the latest available value of ϕ , i.e. the value of ϕ at the end of the last iteration. The linearized source term is integrated over the control volume, which results in

$$S = S_P \phi_P + S_U \quad [3-53]$$

the coefficients are given as

$$S_P = V S^P \quad [3-54]$$

and

$$S_U = V S^U \quad [3-55]$$

3.3.4 Finite Difference Equation

Finite difference equation (FDE) is the algebraic equation when all the discretized terms in the general transport equation are joined together. The general formula of FDE is

$$(a_p - S_p)\phi_p = \sum_{nb} a_{nb}\phi_{nb} + S_u \quad [3-56]$$

As explained before, a_{nb} are the neighboring or link coefficients, thus a two-dimensional form of the discretized equations can be written as follows

$$a_p\phi_p = a_w\phi_w + a_e\phi_e + a_s\phi_s + a_n\phi_n + S \quad [3-57]$$

where **P**, **W**, **E**, **N** are the locations where the values are computed. The finite difference equation is in general nonlinear because the neighboring coefficient themselves are function of ϕ_p , and ϕ_{nb} . The FDE results in a set of coupled nonlinear algebraic equations when formulated for each computational cell. An iterative solution scheme is employed which solves each dependent variable sequentially and repeatedly with the target of improving the solution at each iteration. The iteration process continues until a converged solution is obtained.

3.4 Solution Algorithm

The SIMPLEC algorithm (Semi Implicit Pressure Linked Equations Consistent) proposed by Van Doormaal and Ruthbay (1985) was used in the present computation. This is a guess and correct method on a staggered grid arrangement in which velocities and

pressures are stored at different locations. If velocities and pressures are stored in the same node within the control volume, a highly non-uniform pressure field can act like a uniform field in the discretized momentum equation (Versteeg and Malalasekera 1995). The staggering of the velocity avoids unrealistic behavior of the discretized momentum equation for spatially oscillating pressures and it generates velocities at exactly the locations where they are required for the scalar transport convection-diffusion computations. Hence, no interpolation is needed to calculate velocities at the scalar cell faces (Versteeg and Malalasekera 1995).

The overall solution procedure for the SIMPLEC algorithm is shown in *Figure 3-6*. All parameters in the solution procedure are dictated by the number of times a procedure is repeated, and can be specified by the user. These are the number of iterations, the number of continuity iterations, and in the case of a transient simulation is simply repeated at each time step of a transient simulation. The number of iterations performed is dictated by the overall residual reduction obtained. After an iteration, the program calculates a residual for each variable, which is the absolute value of the difference between the new calculated variable and the old variable divided by the calculated variable each control cell.

$$\mathcal{E}_{res} = \left| \frac{x_{new} - x_{old}}{x_{new}} \right| \quad [3-58]$$

The descriptions of the procedure are as follows:

1. All solution values are initialized. For steady state flows in present computation, the initial condition is only a starting point of the iterative procedure.
2. All terms needed for the u , v , and w momentum equations are evaluated for each cell. The linked coefficients and the source term are calculated. When the terms for all cells are assembled, they form a linear system.

3. The linear systems for the u , v , and w momentum equations are solved using a linear equation solver.
4. The mass imbalance is evaluated and an equation for the pressure correction is posed at each cell. All pressure correction equations are then assembled into a linear system and solved using the chosen linear equation solver.
5. The pressure correction values are applied to the pressure field. The velocity field is corrected by using an algebraic relationship between pressure correction and velocity correction.
6. All other discretized transport equations are solved.
7. Steps 2 to 6 are repeated until convergence is obtained.

CFD-ACE+, a commercial computational fluid dynamics software developed by CFD-Research Corporation (CFD-RC) was used to perform the numerical simulation. This software is comprised of three modules: CFD-ACE+TM, an advanced numerical finite-volume flow solver for structured and unstructured hybrid grids, CFD-GEOMTM, a geometric modeling and grid generation module, and CFD-VIEWTM, a post processing data visualization software module. The computing system was an ATXSTF Mint Professional Gateway personal computer with 520 MB RAM, a Pentium 4 (~1.3 GHz) processor, and a hard drive of 37 GB.

Figure 3-6. Solution flowchart for the SIMPLEC algorithm.

Chapter 4

Computational Methodology

The approach utilized to generate the geometries and the mesh was different for each simulation. The method of applying the fluid properties, boundary conditions, initial conditions, and all other parameters needed to run the computations was defined separately according to the model, whether two- or three-dimensional. The geometries in each simulation consisted of a unique mesh. A structure grid was created on the computational domain for the 2-D annulus, 3-D circular and elliptical co-axial jets, and the coflow jet spreading into the ambient. The domains were divided to insure orthogonality, and smooth transition between interfaces. Grid spacing decreased in the areas of importance to capture the detail of the flow during the computations.

The fluid was air, assumed as an ideal gas with a temperature of 20 °C. The reference pressure (ambient pressure) was 101.325 kPa. The kinematic viscosity of air interpolated from Table A-C-5(a) (pg. 730) corresponding to air from Heat Transfer by Lindon C. Thomas at 20 °C and verified by Fig C.2 (pg. 716) in Fluid Mechanics 9th Edition by Steeter was $1.545 \times 10^{-5} \text{ m}^2/\text{s}$. The specific heat (C_p) and thermal conductivity (K) used in the simulation were 1004 J/kg-K and 0.0263 W/m-k respectively. Density was calculated using the ideal gas law. These properties were inputted as volume conditions to each numerical analysis and are summarized in Table 4-1.

Table 4-1. Properties of air obtained at $T = 20 \text{ }^\circ\text{C}$ and $P = 101.325 \text{ kPa}$.

Density	ρ	PM/RT	kg/m^3
Kinematic Viscosity	ν	1.545×10^{-5}	m^2/s
Specific Heat	C_p	1.004	kJ/kg-K
Thermal Conductivity	K	0.0263	W/m-K

4.1 Annular Design

A two-dimensional analysis of an annulus with varying outer wall profile and varying length was first carried out. In the first analysis, the outer wall varied in curvature while maintaining a fixed length of 10.16 cm (4.0 inches). Fifth, fourth, and third order polynomial curves were used to construct the outer wall of the annulus. Ho and Gutmark (1987), and Wicker and Eaton (1993) used fifth order nozzle profiles while Hussain and Husain (1989) used a third order curves in their nozzle design. A fourth order curve was selected as an intermediate curve between other profiles utilized by other researchers.

The second analysis consisted of varying the length of the annulus and fixing the outer wall profile. For this analysis, a fifth order polynomial outer wall was implemented and the lengths of the curves were 7.62 (3.0 inches), 8.89 (3.5 inches), and 10.16 cm (4.0 inches). The inner wall of the annulus was generated from a fifth degree polynomial curve with a length of 7.62 cm (3.0 inches), and remained fixed for all analysis.

4.1.1 Geometry Generation – Annular Design*

As mentioned above, the profile of the two-dimensional annulus was generated from polynomial curves. The conditions for generating the profiles were adopted from Montiel, Jasso, Castro (2001), a senior design team in the Mechanical and Industrial Engineering Department at UTEP under the direction of Wicker and Choudhuri, in which a zero slope and curvature was imposed on the inlet and exit of the nozzle. A fifth order inner wall remained fixed for all geometries. The outer wall varied from fifth, fourth to third order and also in length. The exit and inlet distances between the inner and outer

* Dimension of geometries throughout the chapter are not in SI units.

wall remained fixed at 0.6062 (1.5397 cm) and 2.348 (5.964 cm) inches respectively *Figure 4-1*). The annuli profiles were constructed by first determining the polynomial equation. The boundary conditions were:

$$\begin{array}{llll} x = 0 & r = 3.799 \text{ inches (9.6495 cm)} & \partial r / \partial x = 0 & \partial^2 r / \partial x^2 = 0 \\ x = \ell & r = 1.0 \text{ inch (2.54 cm)} & \partial r / \partial x = 0 & \partial^2 r / \partial x^2 = 0 \end{array}$$

The length of the outer curves denoted by ℓ , were 4.0, 3.5, and 3.0 inches. Applying the above boundary conditions, the fifth, fourth and third polynomial equations with $\ell = 4$ inches for the outer annular wall are:

$$\text{Fifth Order: } r_{5^{th}} = -0.0164004\ell^5 + 0.1640039\ell^4 - 0.4373438\ell^3 + 3.799$$

$$\text{Fourth Order: } r_{4^{th}} = 0.032801\ell^4 - 0.174938\ell^3 + 3.799$$

$$\text{Third Order: } r_{3^{rd}} = 0.0874688\ell^3 - 0.5248125\ell^2 + 3.799$$

The outer wall profiles for the polynomial curves of varying length are:

$$\ell = 4.0 \quad r_{4.0} = -0.0164004\ell^5 + 0.1640039\ell^4 - 0.4373438\ell^3 + 3.799$$

$$\ell = 3.5 \quad r_{3.5} = -0.03187243\ell^5 + 0.2788838\ell^4 - 0.6507287\ell^3 + 3.799$$

$$\ell = 3.0 \quad r_{3.0} = -0.06911111\ell^5 + 0.5183333\ell^4 - 1.0366667\ell^3 + 3.799$$

The equation of the inner wall, a fifth order curve of 3.0 inches, which remained fixed for all cases is shown below (Wicker and Eaton 1994a).

$$\text{Inner Nozzle: } r_{in} = -.02610617\ell^5 + 0.195796296\ell^4 - 0.3915926\ell^3 + 1.451$$

The method of deriving these polynomial equations is found in Appendix A.

Once the polynomial equations were determined, a spreadsheet with two hundred data points was generated. The data points, saved using an ASCII format, were imported into ThinkdesignTM 6.0, a Computer Aided Drawing (CAD) software to create the curves. Complete procedures for importing the data points into the CAD software are found in Appendix B. The curves generated in ThinkdesignTM 6.0 were saved as an Initial Graphics Exchange Specification (IGES) file. An IGES file allows the transfer of geometry data from one graphical environment to another. IGES, if properly interpreted and implemented, helps an application programmer import relevant information of complex models, generated in a specific CAD environment, for further processing in an analysis environment (Basu and Kumar 1995). The IGES file was then imported into the grid generation software, (CFD-GEOM) to fully utilize the transfer of data.

The computational domain *Figure 4-1* was divided into five or six zones to insure orthogonality, and smooth transition between grids. A structure grid was applied to the geometry. The annulus consisted of twenty-five divisions along the radial direction and an average of ninety-five divisions along the curve. Grid spacing decreased at the exit of the annulus to capture the detail of the flow during the computation.

Figure 4-1. a) Computational domain for the two dimensional annulus. b) Meshed domain of annulus.

4.1.2 Numerical Modeling – Annular Design

A flow and heat solver was used for the annular analysis. Although temperature did not change, a heat solver was used to simulate a realistic flow that took into account temperature. An isothermal flow of 20° C was assumed. All properties previously mentioned were obtained at this temperature. A 0.7569 m/s axial velocity and a 202.650 kPa pressure ($P_e = 2P_i$) were imposed as the inlet condition. The inlet velocity was determined by the volumetric flow rate equation.

$$Q = A\vec{v} \quad [4-1]$$

A 10 m/s velocity was assumed at the exit of the annulus. Knowing the exit velocity and the area, the exit volumetric flow rate was calculated using equation [4-1]. Assuming no losses, and dividing the annulus into infinitesimal divisions, then the volumetric flow rate entering the control volume at x , is equal to the volumetric flow rate exiting the control

volume at $x + dx$ (equation [4-2]). The inlet velocity was determined using equation [4-3] and calculating each volumetric flow rate at $x + dx$, starting from $x = \ell$ to $x = 0$.

$$Q_i = Q_e \quad [4-2]$$

$$A_i \vec{v}_i = Q_e \quad [4-3]$$

The inlet velocities for the annulus of length 3.0, 3.5, 4.0 inches were calculated using equation [4-3]. The average of the three results yielded an axial velocity of 0.7569 m/s. This velocity was used as the inlet boundary condition for the two- and three-dimensional simulations.

A no slip boundary condition was applied on the walls of the annulus.

Table 4-2 shows the boundary and initial conditions used in the two-dimensional annulus simulations. Velocities, pressure, and enthalpy were under-relaxed to improve convergence. Convergence was achieved in 60 seconds at approximately 1500 iteration.

Table 4-2. Boundary and initial conditions applied to the 2-D numeric simulations.

Type	Name	u (m/s)	v (m/s)	P (kPa)	T (K)
<i>Boundary Conditions</i>					
Inlet - Fixed Velocity	In	0.7569	0.0	202.650	293.15
Outlet - Fixed Pressure	Out	---	---	101.325	293.15
<i>Initial Conditions</i>					
	$u = 0.3$ (m/s)			$P = 101.325$ (kPa)	
	$v = 0.0$ (m/s)			$T = 280$ (K)	

4.2 Coaxial Nozzle Design

Three-dimensional numeric simulations of circular and elliptical co-axial nozzles were then performed. These computations were performed to determine the exit annular velocity. The simulation began with designing a 4.0 inch, three-dimensional circular and elliptical nozzle. The three-dimensional jets were restricted to an inlet radius of 3.799 inches, and an exit annular area of 2.655 in². The elliptical nozzle had an aspect ratio of 2:1 (major to minor axis). The outer wall of the elliptical nozzle was also generated by two 5th order polynomial curves, one corresponding to the minor and major axis.

Three-dimensional structured annular meshes were constructed for both the circular and elliptical nozzles. A distinct approach was implemented to develop the 3-D annular meshes for the circular and elliptical nozzles. A detailed description of geometry generation and grid generation for both nozzles is described in the following section. All flow conditions applied to the inlet and exit remained the same for both nozzles. As mentioned before, all fluid properties were determined at 20°C.

4.2.1 Geometry Generation – Coaxial Nozzle Design

The three-dimensional axisymmetric coaxial jet was designed by revolving the fifth order, four inch long, 2-D annulus every 45° as seen in

Figure 4-2. Each revolution provided a surface to construct a mesh. The nozzle consisted of twenty-five divisions along the radial direction, twenty-five divisions on each 1/8 circumference of the circle, and ninety-five divisions along the axial direction. Two three-dimensional annular meshes, a top and bottom half were constructed and then

combined. A three-dimensional mesh of the nozzles' annulus was developed from six meshed surface planes located on the top half of the nozzle. The six surfaces were the outer and inner nozzle walls, the inlet and exit faces, and the two annulus profile (*Figure 4-3*). The bottom half of the nozzle was meshed using the same method. The top half of

the meshed nozzle and a complete mesh are shown in *Figure 4-4*.

Figure 4-2. Wire frame of the circular coaxial nozzle generated by a 45° revolution of the four inch annulus.

Figure 4-3. Surfaces needed to construct the 3D mesh on the top half of the coaxial nozzle.

Figure 4-4. a) Top half meshed nozzle. b) Complete 3-D mesh.

The elliptical nozzle was generated by determining the fifth order polynomial curves corresponding to the major and minor axis plane. The preliminary design of the elliptical profiles was developed by the Montiel, Jasso, and Castro (2001), a senior design team.

The polynomial curves were generated by implementing the same method described in Appendix A. The elliptical nozzle starting from a circular inlet contracted to an elliptical exit with a 2:1 aspect ratio. A radius of 0.7071 and 1.4142 inches in the minor and major axis respectively, ensured an exit area of 2.655 in². The boundary conditions to solve for the two polynomial equations were:

$$\begin{array}{llll} \text{Major Axis Plane:} & x = 0 & r = 3.799 \text{ inches} & \partial r / \partial x = 0 \quad \partial^2 r / \partial x^2 = 0 \\ & x = \ell & r = 1.4142 \text{ inch} & \partial r / \partial x = 0 \quad \partial^2 r / \partial x^2 = 0 \end{array}$$

$$\begin{array}{llll} \text{Minor Axis Plane:} & x = 0 & r = 1.0 \text{ inch} & \partial r / \partial x = 0 \quad \partial^2 r / \partial x^2 = 0 \\ & x = \ell & r = 0.7071 \text{ inch} & \partial r / \partial x = 0 \quad \partial^2 r / \partial x^2 = 0 \end{array}$$

The 5th order polynomial equations implemented to generate the elliptical nozzle were:

$$\text{Major Axis Plane:} \quad r_{maj} = -0.139736\ell^5 + 0.1397360\ell^4 - 0.372629339\ell^3 + 3.799$$

$$\text{Minor Axis Plane:} \quad r_{min} = -0.1811668\ell^5 + 0.1811668\ell^4 - 0.483111467\ell^3 + 3.799$$

Once the polynomial equations were determined, two hundred data points were generated in a spreadsheet, imported to a CAD software, saving the generated curves as an IGES file, and imported into the grid generation software. Appendix B describes the method of importing the curves into the various software packages. An ellipse at an x, y, z location of (4,0,0) with $r_{maj} = 1.4142$ inches and $r_{min} = 0.7071$ inches was first drawn as a reference point to ensure that the two imported curves intersected the ellipse at its major and minor axes. The curve on the major axis plane was revolved in steps of 90° up to 360°. This was done to create the circular inlet of the nozzle. The curve on the minor axis plane was rotated 180°. The revolution of the curve lying on the major axis plane created unwanted curves, one that lied on top of the minor axis curve as seen in *Figure*

4-5. These curves were deleted to obtain the elliptical exit. It was necessary to revolve this curve every 90° to avoid an error when meshing the outer nozzle surface.

Figure 4-5. *Wire frame of the elliptical nozzle.*

An error generated by the grid generation software stated that two points were overlapping, thus prohibiting the creation of a surface mesh on a quarter of the nozzle. This error occurred when the major axis curve was duplicated and rotated 180° , followed by the construction of a three point circle needed to create the inlet surface. The software treats an entity differently when it is rotated or revolved. The rotation command rotates the entity or duplicates it if specified, while the revolution command also rotates the entity, but with attached curves as shown in . The inner nozzle was generated by revolving the curve corresponding to the inner nozzle every 90° .

Figure 4-6. a) *Example of rotation.* b) *Example of revolution*

The elliptical nozzle consisted of twenty-five divisions along the radial direction, fifty divisions on each $\frac{1}{4}$ circumference of the circle, and ninety-five divisions along the axial direction. A three dimensional mesh of the annulus was constructed at every quarter of the nozzle instead of each half due to the non symmetric geometry. The four three-dimensional meshes were then combined into one. The six faces used to construct the three-dimensional mesh were the inner and outer nozzle walls, the inlet and exit faces, and the two annulus profile. One annulus profile consisted of an outer wall generated from the equation corresponding to the major axis plane and the other corresponding to the minor axis plane. Figure 4-7 illustrates the annular three-dimensional mesh on a quarter of the jet and a complete mesh of the elliptical nozzle. There were a total 45,6000 nodes, and 43,3152 cells in the axisymmetric coaxial nozzle. The non-axisymmetric nozzle contained a similar amount of nodes and cells as that of the axisymmetric coaxial nozzle with 465,500 and 442,176 accordingly.

Figure 4-7. a) Three-dimensional mesh on a quarter of the non-axisymmetric coaxial nozzle. b) Complete 3-D mesh on the non-axisymmetric coaxial nozzle.

4.2.2 Numerical Modeling – Coaxial Nozzle Design

A flow and heat solver was applied to the numeric simulation. A no slip boundary condition was imposed on the walls of the annulus. An isothermal flow of 20°C was assumed. Table 4-3 displays the boundary and initial condition applied to the circular and elliptical coaxial nozzle. Each velocity component, pressure, and enthalpy were under-relaxed to improve convergence due to the number of nodes. Convergence was achieved in seven days. To reduce computational time, the numeric simulations were performed in steps continuing each run from the previous to allow for the adjustment of relaxation values. This method reduced convergence to an average of three days as apposed to seven days.

Table 4-3. Boundary and initial conditions applied to the 3-D coaxial nozzle analysis.

Type	Name	u (m/s)	v (m/s)	w (m/s)	P (kPa)	T (K)
<i>Boundary Conditions</i>						
Inlet - Fixed Velocity	In	0.7569	0.0	0.0	202.650	293.15
Outlet - Fixed Pressure	Out	---	---	---	101.325	293.15
<i>Initial Conditions</i>						
	$u = 0.3$ (m/s)		$P = 101.325$ (kPa)			
	$v = 0.0$ (m/s)		$T = 280$ (K)			
	$w = 0.0$ (m/s)					

4.3 Coflow Jet Analysis

A computational domain to accommodate and capture the spreading of a turbulent circular and elliptical coaxial jet was developed. The domain consisted of the outer and inner nozzle profile (circle/circle for axisymmetric coaxial nozzle or ellipse/circle for non-axisymmetric coaxial nozzle), and a structured grid was used for both simulations with the distance between nodes equal to each other. Several geometric shapes were considered. One being a square domain with dimension of 18 x 18 x 12 inches was proposed but discarded due to skewness caused by the transition from a circle to a square. The chosen domain was an enlargement of the exit outer nozzle profiles. The dimensions of the domain were $9r_o$ in the radial direction (where r_o is the exit radius of the outer nozzle in the y and z direction) by 12 inches in the axial direction.

A turbulent solver, in addition to the flow and heat solver was added to this simulation. The $k-\varepsilon$ turbulent computational model was implemented to the jet spreading. Equations [3-41] and [3-42] were used to determine the turbulent kinetic energy and the dissipation rate.

4.3.1 Geometry Generation – Coflow Jet Analysis

A circular domain was generated for the axisymmetric jet. The exit nozzle profiles from the three-dimensional nozzles were copied and pasted into a new drawing area. The domain had a radius of 9.0 inches and a length of 12.0 inches. The first step (*Figure 4-8 a*)) in constructing the domain was to create a point 9.0 inches from the center of the exit nozzles in the radial direction. A duplicated point was translated 12.0 inches in the axial direction. The two points were connected to form a line, and revolved every 45° so that the points created during the revolution coincided with the points on the exit nozzle profile. The next step consisted of extruding the nozzle geometries. The nozzle profiles were extruded 12 inches in the x-direction.

The domain can be considered as three concentric cylinders. Six surfaces must be meshed and then combined to create the 3-D mesh. From the domain, only four surfaces can be obtained, thus, two additional horizontal planes were created to have a total of six surfaces (*Figure 4-8 b*)). The six surfaces are the two horizontal planes, the exit and the inlet of the domain, and the outer and inner wall of the concentric cylinders. The areas within the concentric cylinders were meshed, resulting in six separate 3-D meshes. Due to geometry similarity, all top and bottom half meshes were combined to form one three-dimensional mesh with a single flow volume as seen in *Figure 4-9*.

Figure 4-8. a) Dimension of the circular domain. b) Circular domain with extruded exit jet profiles and horizontal planes needed to create a total of six surfaces.

Several numerical simulations were performed to determine the appropriate grid spacing in the radial and axial direction to achieve grid independence. The $1/8$ circumference of the domain generated by the 45° revolution was divided into 25 divisions to coincide with the division of the inner and outer nozzle exit profiles. A $dx = .05$ was originally proposed, but was increased to 0.5 to decrease computational time and to obtain a rough estimate on the flow field of the jet. As mentioned previously, $dx = dz$. Twenty-four division in the axial direction, and sixteen division in the radial direction result from a $dx = dz = 0.5$ inches. The horizontal plane within the inner most cylinder of the domain was divided into 15 divisions corresponding to the z direction. A domain of $dx = dz = 0.5$ inches resulted in 181,056 cells and 191,256 nodes. The spacing between nodes was then decreased to 0.1 inches. Due to the limitation of computing power, this simulation was unable to run with the present computing system. Increasing the grid spacing to 0.2 inches resulted in 60 divisions along the x -axis and 40 divisions along the z -axis. The

computing system was able to perform this computation. The domain consisted of 792,960 cells and 812,220 nodes. The convergence was achieved after the sixteenth day.

Figure 4-9. *Three-dimensional meshed domain used to capture the jet produced by the axisymmetric coaxial nozzle.*

A 9% difference was obtained when comparing the results between 0.2 and 0.5 inch node spacing. The high difference value discarded the 0.5 spacing between grids. Favorable results were obtained with a node spacing of 0.2 inches, however, choosing a node spacing that would reduce computational time while obtaining the same results as a 0.2 inch node spacing would be desirable. Therefore, a node spacing of 0.3 inches was applied to the domain to obtain results with a minimum percent difference of $dx = dz = 0.2$ in. The horizontal plane of the inner most cylinder was divided into 25 divisions along the z -axis. For this grid spacing, there were 471,744 cells and 487,720 nodes. Convergence was achieved after 24 hours at 1000 iterations. The percent difference ranged from 0.02% to 2.0%. Thus, grid independence was achieved with the given

computational system for the circular jet domain. All future computations dealing with the circular coaxial nozzle were performed using similar grid spacing and domain.

The elliptical domain was constructed similar to the circular domain. The length of the domain from the center of the jet was $9r_{maj}$ and $9r_{min}$, where $r_{maj} = 1.4142$ inches and $r_{min} = 0.7071$ inches. First, an ellipse with a major and minor axis of 12.728 and 6.364 inches was constructed at 90° intervals. The grid generation software allowed for the creation of an ellipse at 90° intervals allowing the domain to coincide with the number of divisions of the outer jet profile. The outer ellipse profile, along with the circular inner nozzle profile was extruded 12 inches. Two planes were then created along the z and y axes as seen in *Figure 4-10*. Again, the planes were needed to obtain the six surfaces in order to create the 3-D mesh. Four 3-D meshes between the outer and inner extruded ellipses and between the cylinder and the extruded inner ellipse were created. Four separate 3-D meshes were created due to the geometry of the domain. The four 3-D meshes created within the same annular area were combined into one mesh. Finally, a plane on the ZX plane was created inside the inner cylinder of the domain and a top and bottom half 3-D mesh were created and combined.

A 0.3 inch grid spacing was implemented in the elliptical domain. The quarter arc of the outer ellipse consisted of 50 division to match the division of the quarter arc of the inner ellipse. When creating a surface mesh, opposite segments must have an equal number of divisions. Therefore, a spacing of 0.3 inches resulted in 38 divisions for any line lying on the y and x -axis. Forty divisions resulted for the lines along the z -axis when $dz = 0.3$ inches. The horizontal plane inside the cylinder was divided into 50 divisions to match the number of divisions in the semi-circle.

Figure 4-10. a) Wire frame of the elliptical domain. b) Three-dimensional meshed domain used to capture the jet produced by the elliptical coaxial nozzle

4.3.2 Numerical Modeling – Coflow Jet Analysis

Circular and elliptical coflow computations with mean velocity ratios ($m = U_o/U_i$) of 0.55 and 1.45 were performed. The $k-\varepsilon$ turbulent model was implemented on this simulation. Equations [3-41] and [3-42] were used to determine the turbulent kinetic energy and the dissipation rate.

$$k = \frac{3}{2} (U_{ref} T_i)^2 \quad [3-41]$$

$$\varepsilon = \frac{C_u^{3/4} k^{3/2}}{l} \quad [3-42]$$

The mean annular exit velocity (U_{ref}) for the axisymmetric jet used to determine k was 22.46 m/s. This velocity was obtained numerically at the exit of the three-dimensional axisymmetric coaxial nozzle. For the elliptical jet, a velocity average between the minor and major axis was calculated to obtain U_{ref} . Three velocity values were average along the major axis and minor axis at the exit of the annulus. Taking the mean of the averaged velocities along the major and minor axis resulted in 21.985 m/s.

A 2% turbulent intensity (T_i) was assumed at the exit of outer jet, while a 1% turbulent intensity based on experimental data was used at the exit of the inner jet. The characteristic length scale (L) was 2.0 cm, which corresponds to the diameter of the inner nozzle. The inner jet velocity was calculated using the ratio relationship and the outer jet velocity of 22.26 m/s and 21.985 m/s obtained from the circular and elliptical coaxial nozzle simulations respectively. Recalling that $l = 0.07L$ and $C_u = 0.09$, and substituting the turbulent intensity and the characteristic length to the appropriate equation, then the values of k and ε for the circular and elliptical coflow jet are obtained. The following table shows the boundary and initial conditions imposed on the domains utilized for the circular and elliptical coflow jet analysis.

Table 4-4. Boundary and initial conditions applied to the domains of the coflow jet simulations.

<i>Boundary Conditions</i>	Name	u	U_i	U_o	k	ε
<i>Circular Coflow Jet</i>	$m = 0.55$	(m/s)	(m/s)	(m/s)	(m ² /s ²)	(J/kg-s)
Inlet	Outer Nozzle	---	---	22.46	0.3027	19.544
	Inner Nozzle	---	40.84	---	0.2502	14.69
	Open Flow	1.00	---	---	0.001	0.004
Outlet - Extrapolated	Out	N/A	---	---	0.3027	19.544
Outlet - Farfield	Bonds	1.00	---	---	0.001	0.004
<i>Circular Coflow Jet</i>	$m = 1.45$					
Inlet	Outer Nozzle	---	---	22.46	0.3027	19.544
	Inner Nozzle	---	15.49	---	0.036	0.8016
	Open Flow	1.00	---	---	0.001	0.004
Outlet - Extrapolated	Out	N/A	---	---	0.3027	19.544
Outlet - Farfield	Bonds	1.00	---	---	0.001	0.004
<i>Elliptical Coflow Jet</i>	$m = 0.55$					
Inlet	Outer Nozzle	---	---	21.985	0.290	18.33
	Inner Nozzle	---	39.97	---	0.2396	13.77
	Open Flow	1.00	---	---	0.001	0.004
Outlet - Extrapolated	Out	N/A	---	---	0.2396	13.77
Outlet - Farfield	Bonds	1.00	---	---	0.001	0.004
<i>Elliptical Coflow Jet</i>	$m = 1.45$					
Inlet	Outer Nozzle	---	---	21.985	0.290	18.33
	Inner Nozzle	---	15.16	---	0.0345	0.7513
	Open Flow	1.00	---	---	0.001	0.004
Outlet - Extrapolated	Out	N/A	---	---	0.290	18.33
Outlet - Farfield	Bonds	1.00	---	---	0.001	0.004
<i>Initial Conditions</i>						
velocity in $x, y, z = 0.0$ (m/s)				$k = 1.0$ (m ² /s ²)		
$P = 101.325$ (kPa)				$\varepsilon = 1.0$ (J/kg-s)		
$T = 280$ (K)						

A low velocity of 1.0 m/s was applied to the boundary condition referred to as *Open Flow* with low values of k and ε were assumed to simulate the movement of air in the ambient. An extrapolated boundary condition that allows the solver to determine the velocity at the exit of the domain was used as the exit condition. Otherwise, the solver adjusts to the inputted value if the outlet boundary condition is fixed at a certain value.

Pressure and velocities are extrapolated from the cell center to the outlet boundary in an extrapolated outlet boundary condition. A zero gradient is assumed at outlet plane boundary. The boundary velocity is then corrected by the pressure correction equation and that value is imposed on the outlet boundary. The values k and ε were set to that of the inlet conditions (*Outer Nozzle*). These values would be used only if flow exited and returned into the domain due to re-circulation.

Another outlet boundary condition named *Bounds* was selected as farfield boundary. This type of boundary assumes to be infinitely large and the gradients of all variables except the pressure variable are zero in the flow direction. Allowing the jet to spread infinitely in the radial direction was the reason for choosing a farfield boundary. The value of k and ε were the same as that used to simulate the ambient air.

Chapter 5

Experimental Facility and Methods

Jet centerline velocity measurements performed by Fernando Jasso and Rogelio Franco, a graduate and undergraduate mechanical engineering students respectively at UTEP, were obtained to validate the numerical results acquired from the coflow jet simulations. The coflow experimental analysis consisted of two velocity ratios, $m = 0.55$ and 1.45 . The circular and elliptical exit annular velocity obtained numerically remained fixed at 22.46 m/s and 21.96 m/s (major axis) respectively. These velocities were to be obtained experimentally at the annular exit plane in order to begin the acquisition of centerline velocities. Centerline velocity measurements were obtained with a single hot-wire probe supported on a two component traverse mechanism.

5.1 Experimental Components

The experimental portion of the project consisted of several components necessary for acquiring velocity measurements from the coaxial jet apparatus. The experimental components included: a coaxial jet apparatus, a flow system, a two component automated traverse system, and two personal computers for data acquisition and control.

5.1.1 Coaxial Jet Rig

The coaxial jet rig developed for the project was positioned 90° with nozzles exits directed vertically downward and supported by 2.00 x 1.00 inch, by 1/16 inch thick steel (4130) rectangular tubing fixture shown in *Figure 5-1*. The fixture placed the nozzle exit 1.23 m (4.00 ft) from the floor, it had a total uninterrupted volume of 1.812 m³ (64ft³), which allowed for full flow development, and permitted access to the interchangeable outer nozzle. Two vertical braces were included in the support fixture to mount and adjust the coaxial jet apparatus.

Figure 5-1. Coaxial jet test rig support frame,

Air was supplied by a 25 hp EP-25 rotary screw compressor manufactured by Ingersoll-Rand's Air Solutions Group. The compressor had the capability to output 900 cfm of air at 125 psi. The dryer was set to run 15 to 20 minutes to remove 97-99% of the humidity prior to turning on the compressor. Once an acceptable humidity was achieved, the compressor was set to run for an additional 15 minutes before acquiring velocity measurements. This allowed the airflow to stabilize. Compressed air flowed into a dryer, model number TS100 manufactured by Ingersoll-Rand's Air Solutions Group, where 97-99% of precipitation was removed. The air flowing in a ¾ inch diameter aluminum pipe flowed through two valves before entering the Stuffer FD100 air filter,

which had the capability of removing moisture and particles. The air filter can remove particles down to 5 microns and 97%-99% of the moisture in the airline. After all particles were removed, the air exited the filter and flowed through another a valve and a pressure gauge before entering the $\frac{3}{4}$ inch main line which divides into three lines to supply the air into the coaxial jet rig. A schematic of the coaxial jet flow system is illustrated in *Figure 5-2*.

Figure 5-2. Schematic of coaxial jet flow system.

An elbow connection fed from a pre-existing $\frac{3}{4}$ inch air supply line located 3.048 m (10.00 ft) from the facility location. The main line coming from the compressor was divided into three individual lines to supply the inner, outer, and smoke generator flow.

One gate valve, one pressure regulator, one needle valve, one flow-meter, one pressure gauge, and one temperature gauge were installed on the flow lines supplying the inner and outer jet flow. Wilkerson pressure regulators, model number R28-06-F000 28A with operating range, 0 to 125 psi were installed to regulate the air flow within the copper pipes supplying the jet with inner and outer flow.

Figure 5-3 illustrates the air supply lines that delivered air to the coaxial jet.

Figure 5-3. Air lines and components that supply air to the inner and outer nozzle.

Air supplied to the inner jet flowed in a 5.08 cm (2 inch) diameter pipe that connected to a 5.08 cm wye (135° bend). The wye then connected to a 5.08 cm schedule 80 PVC pipe to supply the inner flow. Once the inner flow is inside the 20.32 cm (8 inch) scheduled PVC pipe of the annular chamber, a 45° bend redirected the flow vertically downward

traveling 38.1 cm (15 inches) which allowed the flow to develop before entering the 7.62 cm long (3 inch) aluminum inner nozzle (*Figure 5-4*) designed by Wicker and Eaton (1994a).

Figure 5-4. Aluminum 7.62 cm long inner nozzle with 5th order contraction.

Flow entered the annular chamber from the axial manifold. The axial manifold consisted of one, 3.81 cm (1.5 inch) inlet and four, 3.81 cm exit ports. Attached to the exit ports of the axial manifold were four, equal length 3.8 cm diameter hoses connected to four, 3.81 cm diameter scheduled 80 PVC pipes. Twelve inches of flow development occurred before entering the area contraction. The coaxial jet test facility with labeled components is shown in *Figure 5-5*. The coaxial jet allowed for the installation of the circular nozzle with a 5.08 cm diameter or a 2:1 elliptical nozzle at the end of the 20.32 cm (8 inch) scheduled 80 PVC pipe. The nozzles shown in *Figure 5-6* were designed following the specifications described in Chapter 4 and are summarized in Table 5-1.

Figure 5-5. Coaxial jet test facility and enlarged annular chamber.

Figure 5-6. a) Circular nozzle with a 5.08 cm diameter. b) Elliptical nozzle with a 2:1 aspect ratio.

Table 5-1. Inner, circular, and elliptical nozzle specifications.

<i>Inner Nozzle</i>	
Diameter	2.0 cm
Length	7.62 cm
<i>Circular Nozzle</i>	
Diameter	5.08 cm
Length	10.16 cm
<i>Elliptical Nozzle</i>	
Major Axis Diameter	7.184 cm
Minor Axis Diameter	3.592 cm
Length	10.16 cm

5.1.2 Traverse Mechanism

The two-component traverse, model MB4030P40J-S3 (vertical) and MB4030P40J-S6 (horizontal), were powered by Superior Electric synchronous stepper motors (type M062-

LS09) with 200 steps per revolution. The base of the two-component traverse system was placed at the bottom of the jet and supported by wooden beams attached to the frame (*Figure 5-5*). The traverse mechanism allowed for fully automated positioning of the hot-wire probe.

5.1.3 Data Acquisition and Instrumentation Control

Data acquisition and instrumentation control consisted of two desktop computers as seen in *Figure 5-7*. One computer, a Gateway P5-133 X86 Family 5 Model 2 with 64 MB of RAM was used to control the traverse. A LabVIEW program controlled the movement of the axes and allowed movements ranging from 0.001 to 25 inches. The program used to control the traverse allowed the hot wire probe to cover 26.67 square meters. Incremental displacements of the traverse were inputted into the program and were stored in the memory to repeat the same displacement. The second computer, a Gateway P5-166 with 16.0 MB of Ram was used for the calibration of the hot-wire probe. This computer also acquires, processes, and stores the experimental data.

Figure 5-7. *Data acquisition and instrumentation control set up.*

5.2 Experimental Instrumentation

A TSI 1210-20 tungsten hot-wire probe (*Figure 5-8*) was used to obtain velocity measurements of the jet. A thermal anemometer (hot-wire probe) measures fluid velocity by sensing changes in heat transfer from a small, electrically-heated sensor (wire or thin film) exposed to the fluid under study. The heated sensor is held at a constant temperature using an electronic control circuit. The cooling effect resulting from the fluid flowing past the sensor is compensated for by increasing the current flow to the sensor. The magnitude of the current increase needed to keep the temperature constant is directly related to heat transfer and, thus, affects flow velocity. This parameter is available as an output voltage. The technique can be applied to nearly any flow where the flow is not adversely affected by the physical presence of a sensor or where contaminants in the flow cannot contaminate or damage the fragile sensor. It is especially appropriate for applications requiring very fast response to flow changes (high frequency response), and low turbulence measurements([www.tsi.com]).

Figure 5-8. TSI 1210-20 tungsten hot-wire probe sensor (www.tsi.com).

5.2.1 Hot-wire Calibration

The hot-wire probe was calibrated using TSI's ThermalPRO software. The software has the capability to analyze one-, two-, or three-sensor probe data all communicated via an RS-232 interface. The ThermalPRO software serves four basic functions: control, calibration, data acquisition, and data analysis. For control, it directs all operations of the IFA300 Research Constant Temperature Anemometer, which enables full control of all anemometer functions. The IFA300 anemometer is fully automated with a frequency response of 300 kHz or greater, and combines with an A/D board installed in the P5-166 Gateway personal computer used for data acquisition. The software's data analysis includes on-line time history and probability distribution. The software can calculate and display mean velocity and turbulence intensity. Statistical results can be written to an ASCII file for external analysis. The equipment needed to calibrate the hot wire-sensor were:

- TSI Model 1128 Air Velocity Calibrator
- IFA 300 Constant Temperature Anemometer
- Pressure Transducer
- Single-sensor probe support
- Shorting probe
- Thermocouple (Type T)

To start the calibration, the TSI 1128 Air Velocity Calibrator must be connected to the main air line while the IFA 300 Constant Temperature Anemometer and the ThermalPRO software are running. Detailed procedures on how to calibrate the hot-wire probe are found in Appendix C. The table below describes the equipment utilized in the flow system, the calibration of the hot wire probe, and in acquiring velocity measurements.

Table 5-2. Experimental equipment

Equipment	Manufacturer	Model Number	Description
Compressor	Ingersoll-Rand	EP-25	25 hp 900cfm @125 psi
Dryer	Ingersoll-Rand	TS100	Removes 97-99 % of humidity
Traverse (Vertical)	Velmex, Inc	MB2527P40J-S3	
Traverse (Horizontal)	Velmex, Inc	MB4030P40J-S6	
Pressure Regulators	Wilkerson		0-125 psi
Slo-Syn-Motor	Superior Electric	M062-LS09	200 steps/Rev
Hot Wire	TSI	1210-20	Tungsten wire
TSI Calibrator	TSI	1128B	Automated calibrator
IFA 300			Control all anemometer
Constant Temperature	TSI	IFA 300	functions and fully automated
Anemometer System			Frequency response of 300 kHz
Pressure Transducer	MKS Baratron	220D	
Air Filter	Stuffer	FD100	Removes particles >5 microns Removes 97%-99% of moisture

5.3 Experimental Techniques

Once the hot-wire sensor was calibrated, it was attached to the vertical axis of the traverse mechanism and was placed at a 45° from the horizontal to allow air to flow in a smoother path down the probe support. The probe was positioned at the exit plane of the jet. In order to verify that the centerline of the jet did correspond to the center of the exit plane, a string with a weight was hung from the center of the inner nozzle. The hot wire probe was then placed at the center of the inner jet exit plane to see if it coincided with the location of the string. The probe was then lowered with the traverse to see if it would follow the same path as the string. If the probe did not follow the same vertical path as the string, the annular chamber was then adjusted and the process was repeated until achieving the vertical centerline of the jet.

At the jet exit plane, measurements were taken at the center of the inner jet and at the center of the annulus. Measurements were then acquired, processed, and viewed using the IFA 300 ThermalPRO program. Depending on the exit jet velocity result, airflow regulators were adjusted until the desired exit jet velocities of the inner and outer jet were obtained. A 22.46 m/s annular exit jet velocity (U_o) was to the target velocity for the circular outer nozzle and the elliptical nozzle target annular exit velocity could range from $21.96 < U_o < 22.01$ m/s in the major axis.

Table 5-3 displays the experimental annular and inner nozzle exit velocities for the circular and elliptical coaxial nozzles for the corresponding velocity ratios. Inner and outer jet Reynolds numbers is also presented.

Table 5-3. Experimental conditions.

<i>Circular Coaxial Nozzle</i>	<i>Elliptical Coaxial Nozzle</i>
$U_o/U_i = 0.55$	$U_o/U_i = 0.55$
$U_i = 40.50$ m/s	$U_i = 40.12$ m/s
$U_o = 19.59$ m/s	$U_o = 20.02$ m/s
$Re_i = 62,566$	$Re_i = 61,980$
$Re_o = 46,606$	$Re_o = 102,124$
$U_o/U_i = 1.45$	$U_o/U_i = 1.45$
$U_i = 15.50$ m/s	$U_i = 15.35$ m/s
$U_o = 22.91$ m/s	$U_o = 22.13$ m/s
$Re_i = 23,945$	$Re_i = 23,714$
$Re_o = 54,505$	$Re_o = 112,888$

After the desired inner and outer exit jet velocities were obtained, the program controlling the traverse mechanism was set to move in 0.125 inch increments in the downward direction. The first measurement was taken at the center of the jet exit plane. Approximately 5 seconds were required to record the air flow velocity. The hot-wire probe was then positioned 0.125 inches below the previous point and the second measurement was recorded. This process was repeated until the hot-wire probe reached a distance of 30.48 cm (12 inches) from the jet. A total of 96 measurements were obtained.

Chapter 6

Computational and Experimental Results

Chapter 6 is divided into two sections, Section 6-A and 6-B. Section 6-A discusses the computed results from the two- and three-dimensional annular flows. Velocity contours within the annulus are displayed in Section 6-A. The velocity inside the annulus is also described by axial and radial velocity distribution plots.

Section 6-B presents the near-field flow characteristics of a circular and elliptical coaxial jet. The section begins with a discussion on the velocity magnitude and vorticity circulation contours that display the near-field velocity and mixing characteristics of both jets. Experimental centerline velocities at 0.55 and 1.45 velocity ratios are also presented in this section. Section 6-B also contains computed radial velocity distributions, turbulent kinetic energy, and RMS turbulent fluctuating velocity plots for the axis- and non-axisymmetric jets.

Section 6-A

Section 6-A presents the computed annular flow results through two-dimensional annuli and three-dimensional axisymmetric and non-axisymmetric coaxial nozzles. The section describes the effect of curvature and length on the velocity profiles within a two-dimensional annulus and most importantly the velocity distribution exiting the annulus. The velocity profiles within the three-dimensional annular area of the coaxial nozzles are also presented in Section 6-A of Chapter 6.

6-A.1 Two Dimensional Annulus

Radial and axial velocity along with contour diagrams of two-dimensional annuli of varying outer wall and length explain the behavior of velocity inside and at the exit on the annuli. The outer wall consisted of 5th, 4th and 3rd order polynomial curves. The three lengths of $\ell = 7.62$ cm, 8.89cm, and 10.16 cm were varied.

6-A.1.1 Effect of Curvature

The computational results for the annulus of varying curvature are shown in *Figure 6-1*. The axial velocity and the velocity distribution profiles along the radial direction were plotted to explain the effect of curvature on velocity. The axial velocities at different radial locations are plotted in *Figure 6-2*. In all three plots, a steady increase in velocity is observed with a decline in pressure within the annulus. An annulus with a fourth order outer wall profile produces the slowest axial velocity. This is attributed to the minimal

reduction in area as seen in *Figure 6-1*. The axial velocity accelerates more with an annulus of 5th order outer wall as seen in *Figure 6-2*.

The three annulus ducts exited at 5.8 m/s when the velocity was measured at $\frac{1}{4}$ the distance from the inner wall. Comparing the axial velocity at $\frac{1}{4}r$ (the distance from the inner wall) to centerline and $\frac{3}{4}r$, it is observed that a faster velocity is obtained near the outer wall. This results from the bend on the annulus. This behavior agrees with the results reported by Berger, Talbot, and Yao (1983) in which the axial velocity is faster at the inner wall (since the initial profile is that of an inviscid vortex), and the boundary layer is thinner there because of the shorter wall length. Berger, Talbot, and Yao (1983) further discuss that as the point of maximum axial velocity moves forward, the boundary layer is thinned at the outer wall because of the axial acceleration, and the boundary layer is thickened at the inner wall. The thickening of the outer wall causes the flow to slow down as observed at an axial distance of 7.62 cm.

The axial velocity at $\frac{3}{4}$ the distance from the inner wall accelerates the fastest near the exit due to the second bend of the annulus and the reduction of area. Maximum velocity is obtained at approximately $\ell = 8.89$ cm followed by a decrease in velocity resulting from a slight pressure increase.

Figure 6-1. Velocity contours within the annuli of fixed inner wall and varying outer wall.

Figure 6-2. Axial velocities at three different radial locations measured from the inner wall of an annulus.

The velocity distribution as a function of radius at three different axial locations along the annulus is plotted in *Figure 6-3* and *Figure 6-4*. At $\ell = 5.08$ cm, the velocity distribution of all curves have the velocity near the inner wall moving faster than the outer wall. This corresponds to the first bend location. The boundary layer is thinner in this section thus causing the flow to accelerate near the inner wall. The velocity profiles corresponding to the fifth order polynomial curve accelerate the fastest due to the reduction of area. The fourth order is the slowest because of minimal change of area. The location of a second bend in the annulus ($\ell = 7.62$ cm) shows a non-uniform velocity distribution in which the flow near the outer wall accelerates faster. At this location, a uniform velocity distribution for the fourth order polynomial is observed. This results from a no slope change in the annulus. The exit velocity profiles in *Figure 6-4* shows that the velocity near the outer wall is moving faster than the inner wall. The fourth order polynomial curves produce a more uniform exit velocity.

Figure 6-3. Radial u velocity distributions at axial locations of 5.08 cm and 7.62 cm.

Figure 6-4. Radial u velocity distribution at an axial location of 10.16 cm.

6-A.1.2 Effect of Length

Similar flow patterns along the axial and radial directions were obtained when varying the length while fixing the outer and inner wall. The maximum velocity for each annulus was achieved at the second bend as depicted in *Figure 6-5*. A length equal to 7.62 cm results in the fastest exit velocity, but with less uniformity. Uniformity is achieved as length is increased. Uniformity with increasing length is difficult to observe though, however Table 6-1 clearly demonstrates a decrease in percent difference as the length is decreased.

Figure 6-5. Velocity contours within the annuli with varying length.

Figure 6-6. Exit velocity for annuli of varying length.

Table 6-1. Percent difference between velocities near the outer and inner wall of an annulus of varying length.

Towards Center of Annulus ↓ Center of Annulus	% Difference $\ell = 7.62$ cm	% Difference $\ell = 8.89$ cm	% Difference $\ell = 10.16$ cm
	6.40%	6.40%	4.72%
	6.47%	6.47%	4.58%
	6.52%	6.42%	4.56%
	6.27%	6.10%	4.31%
	5.78%	5.64%	3.97%
	5.21%	5.09%	3.56%
	4.57%	4.48%	3.12%
	3.88%	3.82%	2.65%
	3.16%	3.11%	2.16%
	2.40%	2.37%	1.64%
	1.61%	1.60%	1.10%
	0.81%	0.80%	0.55%
	0.00%	0.00%	0.00%

6-A.2 Three-Dimensional Circular and Elliptical Coaxial Jets

The velocity contours for the circular and elliptical jets are displayed in *Figure 6-7*. The velocity is uniform around the circumference of the annulus. The axial velocities (*Figure 6-8*) for the circular and elliptical nozzles at three different radial positions are plotted. The axial velocity of the elliptical jet was separated into the minor and major axis. In all three radial positions, the axial velocity of the circular, minor axis, and major axis are the same up to $\ell = 2.54$ cm. At $\ell = 2.54$ cm the elliptical nozzle, initially with a circular entrance, starts transforming into an ellipse, as seen in *Figure 6-9*. The axial velocity continues to increase without any abrupt changes in flow thus, resulting in a satisfactory nozzle design.

Figure 6-7. Velocity contours within the annulus of a coaxial nozzle with a) circular and b) elliptical outer nozzle.

Figure 6-8. Annular u velocity at three different radial locations measured from the inner wall of the circular or elliptical coaxial nozzles.

Figure 6-9. Outer wall profiles for the circular, elliptical, and inner nozzles.

The velocity distribution in the radial direction (*Figure 6-10*) was plotted to demonstrate the behavior of flow within the annulus of the jet. Referring to *Figure 6-10*, at $\ell = 5.08$ cm, the acceleration of flow near the inner wall results from the first bend of the annulus. The minor axis velocity distribution, having a steeper slope, accelerates the fastest due to the greatest reduction in area as compared to the circular and the major axis. At $\ell = 7.62$ cm the circular velocity distribution exhibits a uniform distribution while the minor and major axis are non-uniform. The velocity approaches the second bend causing the velocity associated with the outer wall to accelerate. The exit velocity of the circular jet is uniform exiting at approximately 22.5 m/s. The elliptical velocity distribution is also uniform with an exit velocity of approximately 22.0 m/s with the minor axis velocity exiting at a slightly faster velocity, due to a smaller exit area.

Figure 6-10. Radial velocity distributions at three different axial locations.

6-A.3 Validation of Numeric Results

The results for the annuli and the coaxial nozzles were verified using the continuity equation stating that

$$\dot{m}_i = \dot{m}_e \quad [6-1]$$

which can be written as

$$(\rho A \vec{v})_i = (\rho A \vec{v})_e \quad [6-2]$$

Replacing density with the equation of state yields

$$\left(\frac{P}{RT} A \vec{v} \right)_i = \left(\frac{P}{RT} A \vec{v} \right)_e \quad [6-3]$$

The inlet and exit temperature remained constant, thus canceling that variable along the universal gas constant of air. Equation [6-3] further reduces to

$$\frac{P_i}{P_e} \left(\frac{A_i}{A_e} \right) \vec{v}_i = \vec{v}_e \quad [6-4]$$

Equation [6-4] was used to calculate the exit velocity that was compared against the computational results.

Recalling from Chapter 4, the exit and inlet pressure were 101.358 kPa and 202.65 kPa respectively. The inlet velocity was set to 0.7569 m/s. For two-dimensional annuli, a difference in height was substituted for the inlet and exit area. The change in height for the inlet of the annulus was 5.964 cm and 1.54 cm for the exit. Substituting these values into Equation [6-4] yielded an exit velocity of 5.86 m/s, which is comparable to the values in *Figure 6-4* and *Figure 6-6*.

The same procedure was performed when determining the annular velocity of the three-dimensional nozzles. The annular inlet and exit annular area for the nozzles were 249.845 cm^2 and 17.13 cm^2 respectively. The inlet to exit pressure ratio was 2.0 and an inlet velocity of 0.7569 m/s was the inlet boundary condition. Applying equation [6-4] yielded an exit velocity of 22.08 m/s which also compares to the results obtained from the computational models as seen in *Figure 6-7*, *Figure 6-8*, and *Figure 6-10*.

Section 6-B

This section discusses computed flow fields of jets issued from circular and elliptical with 0.55 and 1.45 velocity ratios. Velocity contours at different axial positions showing the front view of the jet spreading are discussed. Next, centerline velocities and turbulent kinetic energy plots are discussed. The numeric centerline velocities are compared to the experimental measurements obtained from the coaxial jet test rig. The section concludes with radial plots of u velocity, turbulent kinetic energy, and the fluctuating velocity components at various axial distances.

6-B.1 Velocity Magnitude Contour Planes

The velocity magnitude contour planes start at the center of the jet and move in the radial direction in increments of 0.9525 cm (.375 in). The jet formed by the non-axisymmetric coaxial nozzle contains a minor and major axis plane. The planes of the axisymmetric jet are referred to as the circular XY planes. A visual comparison on the spreading of the jet and its behavior due to different geometry and velocity ratios were performed.

6-B.1.1 Contour Planes for $m = 0.55$

The velocity magnitude contour planes corresponding to $m = 0.55$ are displayed in *Figure 6-11* through *Figure 6-13*. With $m = 0.55$, the inner nozzle for both coaxial nozzles has a higher velocity ($U_i \approx 40$ m/s) than the outer nozzle ($U_o \approx 22$ m/s). The inner and outer flow is clearly recognized in the planes at the center of the jet where the pink contour corresponds to the velocity of the inner nozzle and the light green contour corresponds to the velocity of the annular flow. The axisymmetric coaxial nozzle exhibits a faster centerline velocity because the pink contour maintains its width as it travels downstream. From *Figure 6-13*, it is observed that the major axis plane produces a wider jet. This is due to the greater exit radius when compared to the circular and minor axis diameter. A unique feature on the major axis plane is that at 15.24 cm the jet contracts. This feature is observed by tracing the dark green contour that corresponds to approximately 20 m/s. The minor axis plane also shows contraction and expansion of the jet near the exit of the coaxial nozzle. The contraction of the jet on the major axis plane and the expansion in the minor axis plane corresponds to the early stages of axis switching that an elliptical jet undergoes as it spreads. The contraction of the jet could also be due to the merging of the outer and inner flow, which signifies the beginning of the similarity region.

The next plane corresponds to a radial distance of 0.9525 cm from the center. This plane is inside (close to the wall) the inner nozzle that has a radius of 1.0 cm. This explains why the velocity of the inner flow is still visible (pink and red contours). The minor axis and the circular XY plane look similar, and a decrease in velocity is observed at the center ($x,0$) of the major axis plane.

The magnitude velocity is within 25 m/s to 15 m/s at a radial distance of 1.905 cm from the center of the jet. The circular XY plane has a constant velocity of approximately 20 m/s in the exception of the location near the coaxial nozzle exit. The velocity near the

exit in the minor plane extends further axially, approximately 15.24 cm at a velocity of about 22 m/s than the velocity in the circular XY plane. The major axis plane exhibits the slowest velocity of all three planes. This occurs due to the location of the major axis plane, which is located 0.12 cm away from the edge of the elliptical nozzle in the minor axis direction ($r_{min} = 1.796$ cm).

In the next plane (radial distance of 2.8575 cm from the center), a decay in jet velocity for all three planes is observed. The velocity near the coaxial nozzle exit in the circular XY plane begins to decay below 10 m/s. The minor axis plane contains the fastest velocity magnitude of approximately 20 m/s. This results from the plane being within the annular area. The minor axis plane represents the annular flow exiting through the major axis of the ellipse. The major axis plane only captures the downstream spreading of the jet because the plane is away from the edge of the outer elliptical nozzle.

The last velocity magnitude contour plane is located at 3.81 cm. A velocity of about 7m/s is present at the end of the domain for the circular XY plane. The minor axis plane displays this velocity magnitude throughout the plane. The major axis plane only displays minimum velocity near the end of the domain. This results from the plane being 0.22 cm away from the edge of the nozzle and the fact that the major axis structures are contracting due to axis switching.

Figure 6-11. Magnitude velocity contours on the circular XY planes for $m = 0.55$.

Figure 6-12. Magnitude velocity contours on the minor axis plane for $m = 0.55$.

Figure 6-13. Magnitude velocity contours on the major axis planes for $m = 0.55$.

6-B.1.2 Contour Planes for $m = 1.45$

For $m = 1.45$, the contour planes are shown in *Figure 6-14* through *Figure 6-16*. The inner velocity is approximately 15 m/s and an annular velocity of about 22 m/s is observed. Looking at the planes at the center of the jet, it is observed that the width of the inner flow velocity starts to decrease at about $x = 15.24$ cm. This is most evident on the minor axis plane. The narrowing of the inner jet velocity occurs when the annular flow velocity enters the area of the inner jet. Overall, the jet spreads slightly wider in the major axis plane than in the circular XY plane.

Moving 0.9525 cm from the center, the contour planes are dominated by the maximum velocity that the outer nozzle produces which is approximately 22 m/s. These planes are located at the edge of the inner nozzle, and show the magnitude velocity produced by the annular jet. The circular coaxial nozzle has the strongest velocity field as shown in the pink and red contour colors with the velocity magnitude of the inner jet not present. The presence of the inner jet (the orange region) is more visible in the minor and major axis planes and visually has the same length. The velocity decay near the center ($2.54 \text{ cm} < y < -2.54 \text{ cm}$) is greater in the major axis plane as the jet travels downstream. This decay of centerline velocity is attributed to the mass entrainment caused by the pumping mechanism produced during axis switching.

The next location of 1.905 cm shows the presence of the outer flow near the exit of the circular coaxial nozzle followed by a gradual decrease along the centerline. The minor axis plane demonstrates the dominating annular flow of the elliptical jet since the plane lies within the annular area. The contours in the major axis plane display a velocity range between 12 m/s to 8 m/s. The velocity magnitude is much weaker because this plane is past the edge of the elliptical nozzle. There is a yellow region on this plane, which signifies a higher velocity. This region results from a faster annular velocity.

At 2.8575 cm, the velocity of the jet on the circular XY plane is approximately 10 m/s. This plane captures the spreading of the jet from a axisymmetric coaxial nozzle because the plane is located at the edge of the outer nozzle. The minor axis plane still shows the exit annular velocity (≈ 22 m/s) near the exit of the asymmetric coaxial nozzle. The major axis plane captures the trailing end of the jet.

In the last location, the planes are capturing the downstream spreading of the jet. The axisymmetric coaxial nozzle spreads wider than the asymmetric nozzle in the major axis plane. The major axis plane has a region near the end of the domain resulting from the plane being 0.22 cm away from the edge of the elliptical nozzle. The minor axis plane shows a jet that is spreading with a velocity magnitude of about 8 m/s. Thus, a faster jet is produced in the minor axis plane.

Figure 6-14. Magnitude velocity contours on the circular XY planes for $m = 1.45$.

Figure 6-15. Magnitude velocity contours on the minor axis plane for $m = 1.45$.

Figure 6-16. Magnitude velocity contour on the minor axis plane for $m = 1.45$.

6-B.1.3 Comparison Between $m = 0.55$ and $m = 1.45$

The global magnitude velocity contour planes for $m = 1.45$ are exposed in *Figure 6-17* through *Figure 6-19*. For the comparison process, all contour planes have the same velocity magnitude scale ranging from 0 to 43.19 m/s. The contour planes in *Figure 6-17* through *Figure 6-19* are compared to the planes in *Figure 6-11* through *Figure 6-13*. Comparing the circular XY planes for both velocity ratios, all planes look identical by visual inspection, with the exception of the first two locations. This behavior coincides with the independence of structures with velocity ratios. The ratios do not influence the center structures that cause disturbances in the surrounding air. No change in the surrounding air is seen in the contour velocity planes as the planes move away from the edge of the inner nozzle. The similarity of the planes with the exception of the first two locations is not so obvious in the minor and major axis planes. Careful examination of the major axis plane at 1.905 cm reveals different velocity magnitude contours.

Another comparison is done on the planes at the edge of the outer nozzle. The radial distances from the center are 2.8575 cm, 3.81 cm, and 1.905 cm, which correspond to the circular XY, minor axis, and major axis planes respectively. The velocity spreads wider on the major axis plane because of its wider diameter. The jet is narrower on the minor axis plane, while the width of the circular XY plane lies between the width of the jet on the minor and major axis plane. This is a result from the diameter being between the diameter of the major and minor axis.

Figure 6-17. Global magnitude velocity contours on the circular XY plane for $m = 1.45$.

Figure 6-18. Global magnitude velocity contours on the minor axis plane for $m = 1.45$.

Figure 6-19. Global magnitude velocity contours on the major axis plane for $m = 1.45$.

6-B.2 Vorticity Circulation Contour Planes

The vorticity circulation contour planes are incremented at the same distance (0.9525 cm/0.375 in) from the center of the jet as the velocity magnitude contour planes. The vorticity of a fluid is defined as twice the local angular velocity of a fluid element and is represented mathematically as

$$\omega = 2 \frac{d\Omega}{dt} \quad [6-1]$$

where $\frac{d\Omega}{dt}$ is the angular velocity vector defined as

$$\frac{d\Omega_x}{dt} = 1/2 \left(\frac{dw}{dy} - \frac{dv}{dz} \right) \quad [6-2]$$

$$\frac{d\Omega_z}{dt} = 1/2 \left(\frac{dv}{dx} - \frac{du}{dz} \right) \quad [6-3]$$

$$\frac{d\Omega_y}{dt} = 1/2 \left(\frac{du}{dz} - \frac{dw}{dx} \right) \quad [6-4]$$

for rotation about the x , y , and z axis. Vorticity as a relation of velocity can be written as

$$\omega = \text{curl} \mathbf{V} = \nabla \mathbf{x} \mathbf{V} \quad [6-5]$$

and is a measure of rotational effects.

The contour planes illustrate the rotational effects as the jet spreads into ambient air. A visual comparison on the vorticity circulation of the jet due to different geometry and velocity ratios are performed.

6-B.2.1 Contour Planes for $m = 0.55$

The contour planes depicting the vorticity of the fluid are in *Figure 6-20* through *Figure 6-22* for the circular XY, minor axis, and major axis planes. Two regions where vorticity is the maximum are observed within the center of the jet. The regions correspond to the vortices structures formed by the interaction between the outer and inner flow. These two regions are consistent with the results stated by the investigators discussed in Chapter 2, in which two trains of vortex rings develop in the outer and inner shear layers near the exit of the coaxial nozzle. The outer shear layer expands as expected with the circular outer nozzle. The outer shear layer in the minor axis plane also expands, however, the outer shear layer in the major axis plane expands inward into the inner shear layer. The vorticity produced where the inner flow meets the annular flow extends to the middle of the domain having a constant width for an axisymmetric coaxial nozzle. With an elliptical outer nozzle, the vorticity extends further downstream than that produced by a circular outer nozzle. In the major axis plane, the inner vorticity circulation expands to the end of the domain. The vorticity in the minor axis plane expands and contracts before diminishing. The minor or major axis plane implies an intense and vigorous mixing within the inner and outer shear layers formed by a non-axisymmetric coaxial jet.

At a location of 0.9525 cm from the center, the planes are located near the edge of the inner nozzle. In the circular XY plane, the outer vorticity expands and then remains constant throughout, while the inner vorticity circulation combines into one region. The outer vortices expand and the inner vortices separate into two regions as seen in the minor axis plane. The major axis plane exhibits an expansion inward and then outward for the outer vorticity. The inner vortices expand and break apart at the middle of the jet as observed in the major axis plane.

The circular XY and minor axis planes are located inside the annular area for a location of 1.905 cm. These planes only show the outer vorticity. The vorticity region in the circular XY plane is wider near the nozzle and then remains constant. The annular vorticity in the minor axis plane is thinner near the nozzles exit. The major axis plane captures the edge vorticity produced by the elliptical nozzle because it is located at the edge of the elliptical nozzle.

The next two plane locations capture the vorticity as the jet spreads in the radial direction. The planes contain regions of high vorticity even though they are away from the nozzle's edge. In the last location (3.81 cm) the vorticity region pertaining to the major axis plane is only visible at the downstream location.

Figure 6-20. Vorticity circulation contours on the circular XY plane for $m = 0.55$.

Figure 6-21. Vorticity circulation contours on the minor axis plane for $m = 0.55$.

Figure 6-22. Vorticity circulation contours on the major axis plane for $m = 0.55$.

6-B.2.2 Contour Planes for $m = 1.45$

The first planes analyzed are located at the center of the jet. The vorticity contour planes corresponding to $m = 1.45$ are located in *Figure 6-23* through *Figure 6-25*. The minor and major axis planes have an extremely thin inner vorticity region. The outer region in the circular XY and major axis plane expands inward and outward. The expansion of the vorticity region inward is most noticeable in the major axis plane. The outer region vorticity circulation in the minor axis plane only expands as the jet travels downstream.

The planes located at 0.9525 cm from the center of the jet show the width and length of the vorticity in the inner shear layer because the plane is located 0.0475 cm from the wall of the inner jet. The inner vorticity region is shorter in the major axis plane. In the circular XY plane, the outer vorticity spreads inward up the middle of the domain. The vorticity also spreads inward and nearly combines with the inner region in the minor axis plane. A wider vorticity circulation region is observed in the major axis plane.

At a location of 1.905 cm, the major axis plane has reached the edge of the elliptical nozzle. This plane shows the intensity of mixing between the annular flow and the surrounding air. The other two planes lie within the annular area. The bottom and top vortices are large enough to interact with each other as seen in the region near the exit of the coaxial nozzle in the circular XY plane. The top and bottom structures are not large enough in the minor axis planes to combine. The structures in the minor axis plane combine at a plane location of 2.8575 cm. The circular XY plane has reached the edge of the circular nozzle at a location of 2.8575 cm. This plane shows the width of the vorticity structures when the annular flow interacts with the surrounding air.

The last plane location is capturing the remains of the vorticity width in the case of the circular XY and the major axis plane. The major axis plane illustrates the remains of the

structures in the downstream location of the jet. The minor axis plane is now at the edge of the elliptical nozzle and shows the intensity of mixing between the annular flow and the surrounding air.

Figure 6-23. Vorticity circulation contours on the circular XY plane for $m = 1.45$.

Figure 6-24. Vorticity circulation contours on the minor axis plane for $m = 1.45$.

Figure 6-25. Vorticity circulation contours on the major axis plane for $m = 1.45$.

6-B.2.3 Comparison Between $m = 0.55$ and $m = 1.45$

Comparing the circular XY planes, it is observed that the outer vorticity region at the planes located at the center of the jet is the same for $m = 0.55$ and $m = 1.45$, which agrees with the finding of Ko and Au (1985). The dependence of the inner structures on the velocity ratios is observed on the circular XY planes at 0.9525 cm. A $m = 0.55$ value produces a wider inner structure which spreads further downstream than with $m = 1.45$. Outer vortex structures grow larger and combine slightly further downstream with a ratio of $m = 1.45$. This is seen at a location of 1.905 cm. Overall, an $m = 1.45$ produces a wider vorticity region thus implying a more vigorous mixing of air.

The next comparison is between the minor and major planes. The outer structures in the minor and major axis planes vary with the change in velocity ratios unlike the structures forming from an axisymmetric coaxial nozzle. Looking at the planes located at the center, the outer vorticity region is wider with $m = 1.45$. Thus, the outer structures are dependent on m unlike the axisymmetric coaxial nozzles. The inner vorticity region extends further downstream and is wider for $m = 0.55$. In general, for an asymmetric coaxial nozzle a velocity ratio of 1.45 creates a wider vorticity region, thus more mixing is achieved with this ratio.

6-B.3 Spreading Jet Contours

The next contours presented are of a frontal view of a jet spreading. These contours represent the axial velocity component and are displayed in *Figure 6-26* through *Figure 6-29*. The contours start at the coaxial nozzle exit and are incremented 5.08 cm until reaching the end of the domain. Again, a comparison is done between the axis- and non-axisymmetric coaxial nozzles at the two velocity ratios.

6-B.3.1 Spreading Jet Contours for $m = 0.55$

The velocity contours produced by the axisymmetric coaxial nozzle clearly show the jet boundary expanding as it travels downstream. As the jet expands, the velocity decays as observed in the inner jet velocity contours. The axial velocity contours for a non-axisymmetric coaxial nozzle behaves differently due to the elliptical outer nozzle. The contour profiles maintain their elliptical and inner circular shape up to 10.16 cm. A deformation is observed at $x = 15.24$ cm. The minor axis is expanding while the major axis is shrinking. The velocity in the region of the inner nozzle transforms to an elliptical contour as seen from $x = 20.32$ cm to $x = 30.48$ cm. A near symmetrical velocity contour is seen in the last contour plot with an elliptical contour at the center of the jet. The symmetrical contour signifies the mid point in elliptical axis switching. As the velocity contours approach symmetric, the area dominated by the inner flow converts to an elliptical contour with the minor and major axis oriented in the same direction as the elliptical nozzle.

6-B.3.2 Spreading Jet Contour for $m = 1.45$

As with $m = 0.55$, the velocity contours corresponding with the circular coaxial jet exhibit an expansion as the jet travels downstream. As the jet expands, the annular flow velocity moves into the center of the jet. This occurs because $U_o > U_i$, which causes the center of the jet to maintain a velocity of 15 m/s as seen in the last contour plot. This implies that the outer velocity intersects the inner shear layer of the jet.

The elliptical velocity contour maintains its shape up to $x = 10.16$ cm. A noticeable distortion in the minor and major axis is observed at $x = 15.24$ cm. At this location, there are two regions above and below the z-axis that look like a semi-ellipses. The velocity produced from the inner nozzle is still present at this location. The jet continues to spread approaching symmetry ($x = 30.48$ cm) and the two semi-ellipse transform to circles. The two circles represent the region with the maximum velocity surrounded by a region below 15m/s. The two circular regions above and below the z-axis imply that the annular flow intersects the inner shear layer but does not merge with the inner flow.

6-B.3.3 Comparison Between $m = 0.55$ and $m = 1.45$

The jet formed from the axisymmetric coaxial nozzle undergoes a widening as it travels downstream. Which value of m causes a wider jet can not be determined from the contours since the plots do not have the same dimensions. A different value of m causes distinct velocity regions near the center of the jet exiting from a non-axisymmetric coaxial nozzle. This is observed starting at $x = 15.24$ cm. Both values of m cause the elliptical contours at $x = 0$ cm to reach symmetry at $x = 30.48$ cm. The first half of axis switching is captured by these plots.

Axisymmetric Coaxial Nozzle

Non-Axisymmetric Coaxial Nozzle

Figure 6-26. Spreading jet velocity contours of an axisymmetric and non-axisymmetric coaxial nozzle at various axial distances when $m = 0.55$.

Axisymmetric Coaxial Nozzle

Non-Axisymmetric Coaxial Nozzle

$x = 20.32 \text{ cm}$

$x = 25.4 \text{ cm}$

$x = 30.48 \text{ cm}$

Figure 6-27. Continued spreading jet velocity contours of a axisymmetric and non-axisymmetric coaxial nozzle at various axial distances when $m = 0.55$.

Axisymmetric Coaxial Nozzle

Non-Axisymmetric Coaxial Nozzle

$x = 0 \text{ cm}$

$x = 5.08 \text{ cm}$

$x = 10.16 \text{ cm}$

$x = 15.24 \text{ cm}$

Figure 6-28. Spreading jet velocity contours of an axisymmetric and non-axisymmetric coaxial nozzle at various axial distances when $m = 1.45$.

Figure 6-29. Continued spreading jet velocity contours of an axisymmetric and non-axisymmetric coaxial nozzle at various axial distances when $m = 1.45$.

6-B.4 Centerline Velocity and Turbulent Kinetic Energy

Experimental centerline velocities were gathered from the coaxial jet test facility and compared to the numeric results. This was the only form of validation performed. *Figure 6-30* and *Figure 6-32* compare the experimental and computational results for $m = 0.55$ and $m = 1.45$ for the two coaxial nozzles. Dimensionless plots of centerline velocity are displayed in *Figure 6-31* and *Figure 6-33*. The centerline turbulent kinetic energy plots for both velocity ratios were only obtained numerically and are displayed in *Figure 6-34* through *Figure 6-37*.

6-B.4.1 Centerline Velocity for $m = 0.55$

It is observed that the numeric results coincide with the experimental results up to 25.0 cm. Before 25.0 cm, the centerline of the jet is dominated by the velocity of the inner nozzle, which is approximately 40 m/s for both cases. Past 25.0 cm, the experimental centerline velocity starts to decrease faster than the computed values. A greater decline in centerline velocity is viewed with the non-axisymmetric coaxial nozzle.

The centerline velocity of the elliptical coaxial nozzle declines more due to a greater mass entrainment from the surrounding air. The increase of mass inside the jet results from the pumping mechanism (axis switching) caused by the contraction of the major axis and the expansion of the minor axis as seen in the spreading jet contour plots. As the mass within the jet increases, the velocity must decrease to conserve momentum.

The percent difference between the experimental and numeric results is tabulated in Table 6-2. For the axisymmetric coaxial nozzle, a good agreement between the

experimental and computational results is observed. The percent differences between the experimental and numerical results are higher for the elliptic coaxial nozzle than the circular coaxial nozzle. The percent differences increase as the jet travels downstream for both cases.

Table 6-2. Percent difference between the experimental and numerical centerline velocities when $m = 0.55$ for an axis- and non-axisymmetric coaxial nozzle.

Axial Distance (cm)	Circular Coaxial Nozzle	Elliptic Coaxial Nozzle
2.54	0.14	0.21
5.08	0.25	0.06
7.62	0.22	0.47
10.16	0.40	0.20
12.70	0.44	0.16
15.24	0.93	0.94
17.78	1.20	0.85
20.32	1.35	1.49
22.86	1.16	2.78
25.40	1.93	5.70
27.94	2.01	9.58
30.48	4.61	9.17

6-B.4.2 Centerline Velocity for $m = 1.45$

For a velocity ratio of 1.45, the measured and computed centerline velocity agrees fairly well. The centerline velocity increases as the jet moves downstream. A greater increase is observed with the non-axisymmetric (elliptic) coaxial nozzle. The computational results also reveal an increase of centerline velocity at 23 cm. The percent differences between the experimental and numerical results are listed in Table 6-3.

Table 6-3. Percent difference between the experimental and numerical centerline velocities when $m = 1.45$ for an axis- and non-axisymmetric coaxial nozzle.

Axial Distance (cm)	Circular Coaxial Nozzle	Elliptic Coaxial Nozzle
2.54	0.13	0.81
5.08	0.08	1.04
7.62	0.15	0.41
10.16	0.88	1.62
12.7	0.09	1.64
15.24	0.52	1.65
17.78	1.71	1.33
20.32	2.03	1.95
22.86	1.78	2.38
25.4	1.73	0.72
27.94	4.43	4.91
30.48	9.93	7.16

The increase of centerline data results from the merging of the outer flow to the inner flow because $U_o > U_i$. For $m = 1.45$, $U_o \approx 22$ m/s and $U_i \approx 15$ m/s for both scenarios. The experimental results show an increase of 15 m/s to 22 m/s, thus the merging of the two velocity streams occurs closer to the nozzles exit. The numerical results show a merging occurring further downstream which supports the spreading jet contours.

Figure 6-30. Experimental and computational centerline velocity at $m = 0.55$.

Figure 6-31. Dimensionless experimental and computational centerline velocity at $m = 0.55$.

Figure 6-32. Experimental and computational centerline velocity at $m = 1.45$.

Figure 6-33. Dimensionless experimental and computational centerline velocity at $m = 1.45$.

6-B.4.3 Turbulent Kinetic Energy

The centerline turbulent kinetic energy is zero up to 25.0 cm followed by a sharp increase. The non-axisymmetric coaxial nozzle produces a greater centerline turbulent kinetic energy than the axisymmetric coaxial nozzle. For $m = 0.55$, the turbulent kinetic energy at $x = 30.48$ cm corresponding to the elliptical coaxial nozzle is $7.81 \text{ m}^2/\text{s}^2$ compared to $3.58 \text{ m}^2/\text{s}^2$ for the circular coaxial nozzle. The centerline turbulent kinetic energy of the non-axisymmetric coaxial nozzle is $0.646 \text{ m}^2/\text{s}^2$ greater than the turbulent kinetic energy of the axisymmetric coaxial nozzle at $x = 30.48$ cm. This implies a more turbulent mixing with an outer elliptical nozzle. Overall, a velocity ratio of $m = 0.55$ produces more turbulent kinetic energy than with a velocity ratio of $m = 1.45$. This could be attributed to the faster velocity produced by the inner nozzle when $m = 0.55$.

Figure 6-34. Centerline turbulent kinetic energy for an axisymmetric and non-axisymmetric coaxial nozzle for $m = 0.55$.

Figure 6-35. Dimensionless centerline turbulent kinetic energy for an axisymmetric and non-axisymmetric coaxial nozzle for $m = 0.55$.

Figure 6-36. Centerline turbulent kinetic energy for an axisymmetric and non-axisymmetric coaxial nozzle for $m = 1.45$.

Figure 6-37. Dimensionless centerline turbulent kinetic energy for an axisymmetric and non-axisymmetric coaxial nozzle for $m = 1.45$.

6-B.5 Radial Velocity Distribution

The radial velocity distribution plots were taken in increments of 5.08 cm. The radial plots from the axisymmetric coaxial nozzle were obtained in the XZ and XY planes. From *Figure 6-38* the velocity distributions are symmetrical indicating adequacy in numerical output. For all other radial plots concerning the axisymmetric coaxial nozzle, only the XY plane is considered. The radial velocity distribution of the non-axisymmetric coaxial nozzle is divided into the major and minor axis planes.

The velocity distribution plots for $m = 0.55$ are shown in *Figure 6-39* and the dimensionless graphs in *Figure 6-40*. A top hat exit profile forms at the annular and inner nozzle exit as observed in all three planes. As expected, the major axis creates a wider jet due to the major axis diameter being greater than the other diameters. The merging of the outer and inner flow of the circular coaxial jet occurs at 10.16 cm. Merging occurs earlier ($x = 50.8$ cm) in the minor axis plane and further downstream in the major axis plane ($x = 20.32$ cm).

An opposite exit velocity profile from that of $m = 0.55$ in which the outer flow is faster than the inner flow is observed when $m = 1.45$. The development of radial velocity distributions is seen in *Figure 6-41* and the dimensionless plots in *Figure 6-42*. The merging of the annular and inner flow only occurs in the minor axis plane at $x = 30.48$ cm. Merging occurs when the outer flow contracts inward into the inner flow region.

Figure 6-38. Evolution of radial u velocity exiting from the circular coaxial nozzle with $m = 0.55$

Figure 6-39. Radial velocity distributions in the circular XY, minor axis, and major axis planes for velocity ratio of 0.55.

Figure 6-40. Dimensionless radial velocity distributions for $m = 0.55$.

Figure 6-41. Radial velocity distributions in the circular XY, minor axis, and major axis planes for velocity ratio of 1.45.

Figure 6-42. Dimensionless radial velocity distributions for $m = 1.45$.

6-B.6 Radial Turbulent Kinetic Energy

Minimal turbulent kinetic energy (k) is found at the exit of the coaxial nozzles. The scale has been enlarged at the exit to magnify the k radial plots. For $m = 0.55$ (*Figure 6-43*, dimensionless graphs *Figure 6-44*), the exit radial profiles look the opposite of the exit velocity profiles, in which the annular region extends farther than the inner region. Greater concentration of turbulent kinetic energy is found away from the center of the jet. Two outer and two inner symmetrical peaks are observed in the circular XY, minor axis, and major axis planes. The major axis plane exhibits these peaks up to $x = 20.32$ cm, while the peaks are only present at $x = 5.08$ cm in the minor axis plane. The circular XY plane maintains these peaks up to $x = 10.16$ cm. The maximum point of the inner peaks is located at $r = 1.0$ cm from the center. This is the same radial distance of the inner nozzle. The maximum point of the outer peaks is located near the edge of the outer nozzle. These peaks occur at the inner and outer shear layers. These peaks of high k eventually merge downstream causing an increase in the turbulent kinetic energy. The increase is caused by an increase in turbulent intensity which results from the vigorous mixing downstream of the jet.

The exit radial profile of the turbulent kinetic energy also resembles the profile of the exit radial velocity for $m = 1.45$ (*Figure 6-45*, dimensionless graphs in *Figure 6-46*). The two inner peaks are not as obvious when $m = 1.45$. The maximum point of the outer peaks occurs at the edge of the outer nozzle. The turbulent kinetic energy is the maximum at about $x = 5.08$ cm and then decreases as the jet travels downstream. The merging between the outer peaks also occurs in the downstream direction. A wider region of turbulent kinetic energy is observed in the circular XY and major axis plane.

6-B.7 Radial Fluctuating Velocities

There is no significant change between u' , v' , w' in each plane at the same axial distance for both velocity ratios. Therefore, only u' is presented in *Figure 6-47* through *Figure 6-50* along with the dimensionless plots. The plots at $x = 0$ cm have been magnified to see the exiting u' . Minimum velocity fluctuation is observed at the exit of the jet. Peaks similar to the turbulent kinetic energy peaks are also visible. These peaks represent u' in the shear layers caused by the inner and outer flow. Fluctuation of u' velocity is minimal, less than 0.5 m/s at the center of the jet to $x = 25.4$ cm. The disappearing of the peaks occurs where the inner and outer axial velocity merge (refer to *Figure 6-39*). The merging of the two peaks occurs downstream of the exit. The major axis plane has wider area rms axial velocity when compared to the other planes.

Minimal velocity fluctuations also occur at the exit plane for $m = 1.45$. The fluctuating velocities are less than 0.5 m/s. The inner peaks are smaller with $m = 1.45$, implying that less mixing occurs in the inner shear layer (edge of the inner nozzle). The axial fluctuating velocities increase further away from the center of the jet signifying more disturbance in the surrounding air. The maximum u' is seen in the peaks formed near the wall of the outer nozzle corresponding to the outer shear layer. The merging of the u' produced by the merging of the annular flow occurs at $x = 30.48$ cm. The wider region of u' is produced by the major axis plane, staying below 3.5 m/s.

Figure 6-44. Dimensionless turbulent kinetic energy radial plots for $m = 0.55$.

Figure 6-45. Turbulent kinetic energy radial distribution plots in the circular XY, minor axis, and major axis planes for velocity ratio of 1.45.

Figure 6-46. Dimensionless turbulent kinetic energy radial plots for $m = 1.45$.

Figure 6-47. u fluctuating velocity radial distribution plots in the circular XY , minor axis, and major axis planes for velocity ratio of 0.55.

Figure 6-48. Dimensionless u' radial plots for $m = 0.55$.

Figure 6-49. u fluctuating velocity radial distribution plots in the circular XY, minor axis, and major axis planes for velocity ratio of 1.45.

Figure 6-50. Dimensionless u' radial plots for $m = 1.45$.

Chapter 7

Conclusions and Recommendations

An investigation of the near-field flow characteristics of an elliptical coaxial jet with 0.55 and 1.45 velocity ratios was presented in this report. A numerical study on laminar flow through two-dimensional annuli of different outer and inner wall contours and lengths were performed. The understanding of the flow within the two-dimensional annulus led to the development of a three-dimensional circular and elliptical coaxial nozzle. The coaxial nozzles were restricted to an inlet radius of 9.64946 cm (3.799 inches) and an exit area of 17.13 cm² (2.655 in²), with the elliptical nozzle having 2:1 aspect ratio. An investigation of flow inside the three-dimensional annulus of the coaxial nozzles was then conducted. The final step included computational and experimental near flow field investigations of an elliptical jet.

The near-field behavior of elliptic coaxial jet reveals some important information regarding the jet structure which can be tailored to control particle dispersion. For instance, the study determined that unlike axisymmetric coaxial jet, the outer structures of the elliptic coaxial jet change with the velocity ratio (Figures 6-26 to 6-29). This feature could make the burner assembly more flexible to produce favorable aerodynamic fields for a range of particle loading by controlling and organizing jet vortical structures. Since, NO_x conversion efficiencies are highly sensitive to particle loading a better control over the local particle aerodynamics will certainly enhance the scope of NO_x reduction for low NO_x burners.

The specific conclusions drawn from the different parts of the project are discussed in following sections:

7.1 Two-Dimensional Annulus

Data obtained from the annular design was employed to develop the axisymmetric and non-axisymmetric coaxial nozzle. The following conclusions were drawn from the results obtained for a two dimensional annulus.

- i. The velocity accelerates near the wall where the bend is located. Acceleration near the bend results from the thinning of the boundary layer.
- ii. The maximum velocity regardless of length or polynomial order occurs at the second bend near the outer wall. The maximum velocity at this location for all annuli is approximately 7.0 m/s.
- iii. An annulus with a fifth order polynomial outer wall accelerates flow faster than a fourth or third order polynomial outer wall.
- iv. Uniformity in exit velocity is achieved when length is increased and the outer wall remains fixed.
- v. Validation by the continuity equation yields an exit velocity of 5.86 m/s, which is comparable to the numerical values ranging from 5.81 m/s to 5.97 m/s.

7.2 Coaxial Nozzle Design

The concluding remarks for the three-dimensional annular flow of the circular and elliptical coaxial nozzles are as follows.

- i. The axisymmetric and non-axisymmetric coaxial nozzles exhibit an increase in annular velocity without any abrupt changes in the flow.

- ii. The axial velocities within the annulus of the circular, minor axis, and the major axis are the same up to $\ell = 2.54$ cm. Beyond this point, the elliptical nozzle with a circular entrance transforms to an ellipse.
- iii. Annular exit velocity is approximately 22.0 m/s for both coaxial nozzles.
- iv. Validation by applying the continuity equation yields an exit annular velocity of 22.08 m/s for both nozzle geometries. This velocity is comparable to the computation results that range from 22.46 m/s (circular outer nozzle) to 21.94 m/s (in minor axis).

7.3 Coflow Jet Analysis

The results obtained from the coflow jet analysis the most useful in determining the different mixing behavior and velocity profiles between an axisymmetric and non-axisymmetric coaxial nozzle. For the planes located at the center of the jet:

- i. The jet exhibits a contraction on the major axis plane and the expansion in the minor axis plane when $m = 0.55$. This corresponds to the early stages of axis switching or the merging of the outer and inner flow, which signifies the beginning of the similarity region.
- ii. The width of the inner flow velocity starts to decrease at about $x = 15.24$ cm for the planes located at the center of the jet for $m = 1.45$. The narrowing of the inner jet velocity occurs when the annular flow velocity enters the area of the inner jet.

All planes look identical, by visual inspection, with the exception of the first two locations for both velocity ratios. This behavior coincides with the independence of structures with velocity ratios. The ratios do not:

- i. Influence the center structures that cause disturbances in the surrounding air.

- ii. Have an impact on the surrounding air as seen in the contour velocity plane as the planes move away from the edge of the inner nozzle.

For the planes at the edge of the outer nozzle (radial distances from the center are 2.8575 cm, 3.81 cm, and 1.905 cm corresponding to the circular XY, minor axis, and major axis planes respectively):

- i. Velocity spreads wider on the major axis plane because of its wider diameter.
- ii. The jet is narrower on the minor axis plane, while the width of the circular XY plane lies between the width of the jet on the minor and major axis plane.

The vorticity circulation planes have the following conclusions. In the circular XY plane:

- i. An $m = 0.55$ value produces a wider inner structure which spreads further downstream than with $m = 1.45$.
- ii. The outer vortex ring grows larger and combines slightly further downstream with a ratio of $m = 1.45$.
- iii. A velocity ratio of $m = 1.45$ produces a wider vorticity region thus implying a more vigorous mixing of air.

Looking at the minor and major axis plane:

- i. The outer structures vary with the change in velocity ratio unlike the structures forming from an axisymmetric coaxial nozzle.
- ii. The inner vorticity region extend further downstream and are wider for $m = 0.55$.
- iii. A velocity ratio of $m = 1.45$ creates a wider vorticity region, thus more mixing is achieved with this ratio.

The next set of conclusions comes from the spreading jet contours, which represent a frontal view as the jet travels downstream. As expected, the contours formed by the

axisymmetric coaxial nozzle expand in the downstream direction. The non-axisymmetric coaxial nozzle exhibits a different behavior than the symmetric coaxial nozzle. For both velocity ratios pertaining to the elliptical coaxial nozzle:

- i. The elliptical velocity contour maintains its shape up to $x = 10.16$ cm.
- ii. A noticeable distortion in the minor and major axis is observed at $x = 15.24$ cm.
- iii. Elliptical contours at $x = 0$ cm reach symmetry at $x = 30.48$ cm.
- iv. The first half of axis switching is captured by these plots.

For $m = 0.55$:

- i. The area dominated by the inner flow converts to an elliptical contour with the minor and major axis oriented in the same direction as the elliptical nozzle. This occurs at $x = 30.48$ cm.

For $m = 1.45$:

- i. Two semi-ellipses form above and below the z -axis at $x = 15.24$ cm. The two semi-ellipses transform to two circles further downstream. Thus implying that the annular flow intersects the inner shear layer but does not fully merge as with the inner nozzle exit velocity.

The next set of conclusions comes from the centerline velocity. The centerline velocity was obtained numerically and experimentally. For $m = 0.55$:

- i. The numeric results coincide with the experimental results up to 15.0 cm for both coaxial nozzles.
- ii. The centerline velocity for the non-axisymmetric coaxial nozzle declines more due to a greater mass entrainment from the surrounding air caused by the

contraction of the major axis and the expansion of the minor axis as seen in the spreading jet contour plots.

- iii. The percent differences between the experimental and numerical results are below 2 % for a distance of 15.24 cm.

For $m = 1.45$:

- i. Centerline velocity increase in the downstream direction. A greater increase is observed experimentally. This increase results from the intersection of the outer flow with the inner flow.
- ii. There was a poor agreement with the experimental and computational data with percent differences being above 10 %.

Centerline turbulent kinetic energy was also obtained and plotted for both velocity ratios. The centerline turbulent kinetic energy increases past 25.0 cm. The centerline turbulent kinetic energy is:

- i. Greater for $m = 0.55$ than for $m = 1.45$.
- ii. $0.646 \text{ m}^2/\text{s}^2$ greater a non-axisymmetric coaxial nozzle for $m = 0.55$. This implies a more turbulent mixing with an outer elliptical nozzle and a faster inner velocity.

In regards to the radial velocity distribution plots:

- i. The major axis creates a wider jet due to the major axis diameter being greater than the other diameters.
- ii. The merging of the outer and inner flow occurs at $x = 10.16 \text{ cm}$ in the circular XY plane when $m = 0.55$.
- iii. Merging occurs earlier ($x = 50.8 \text{ cm}$) in the minor axis plane and further downstream in the major axis plane ($x = 20.32 \text{ cm}$).

- iv. The merging of the annular and inner flow only occurs in the minor axis plane at $x = 30.48$ cm when $m = 1.45$.

The turbulent kinetic energy turbulent radial plots reveal:

- i. Two inner and outer peaks at the location of the inner and outer nozzle radii when $m = 0.55$.
- ii. The downstream merging of the inner and outer peaks which cause an increase in the turbulent kinetic energy when $m = 0.55$.
- iii. Maximum turbulent kinetic energy is reached at $x = 5.08$ cm followed by a decrease as the jet travels downstream for $m = 1.45$.

The fluctuating radial velocity plots reveal:

- i. Minimal, less than 0.5 m/s fluctuating velocity at the center of the jet up to $x = 25.4$ cm for both velocity ratios.
- ii. Inner and outer peaks where the shear layers form.
- iii. Larger inner peaks for $m = 0.55$ and smaller inner peaks with $m = 1.45$, implying that less mixing occurs in the inner shear layer for this ratio.
- iv. An increase in the rms axial velocity downstream of the jet where fluctuations are greater for velocity ratio of 0.55.

Overall, a non-axisymmetric coaxial nozzle with a ratio of $m = 1.45$ produces more intense mixing. The minor and major axis planes have distinct flow characteristic, which creates more mixing. The outer vorticity region is wider and large structures merge downstream. This behavior is favorable for mixing applications.

7.4 Practical Implications

The present study provides information on the near-field mixing dynamics of single phase coflow jets issued from an elliptic coaxial nozzle. This information will be beneficial to tailor the use of elliptic nozzles for controlling particle dispersion in coal laden flow. For a low NO_x pulverized coal combustor system, particle dispersion dominates the aerodynamic NO_x abatement process. In near burner location for a wide range of oxygen levels, generally the volatile-N species are completely converted to NO_x for particles, which are completely dispersed in an oxidizing environment. Even though the film may contain substantial amounts of hydrogen and hydrocarbon fragments only an insignificant amount of volatile-N is reduced to N_2 within the volatile frame. On the other hand although benign nitrogen species cannot break through the flames surrounding isolated particles, nitrogen species in fuel rich clouds are not necessarily converted into NO_x . Thus it is believed at near burner region maintaining aggregate particle clouds, as on the vortex caps, should lower NO_x levels. The distinct features observed in shear layer development of elliptic coaxial jets are due to the presence of near-field longitudinal vortices (Figures 6-26 to 6-29 and Figures 6-38 to 6-42). This flow feature should be able to create a favorable near-burner aerodynamics to maintain a fuel-rich environment for low NO_x combustion. Thus, it may be possible to use an elliptic coaxial nozzle in low NO_x burner to further reduce NO_x emission without increasing CO production.

7.5 Recommendations for Future Work

Although the present pilot project demonstrates the possibility of elliptic coaxial jets as a means of aerodynamics NO_x abatement in Low NO_x burner, a more comprehensive study is required to understand its effectiveness in complex particle laden flows. Future study involving measurements of particle distributions at different particle loadings and

velocity ratios is recommended to improve the understanding of particle flow interaction in elliptic coaxial jets and to make this novel concept practical.

7.6 Research Infrastructure, Research Training and Education

The proposed project impacted the undergraduate and graduate mechanical engineering (ME) program in a variety of ways. As part of the new ABET accreditation criteria, the UTEP undergraduate ME curriculum has been revised to include extensive experimentation. The coaxial jet facility and other infrastructure developed through this project are being used in: (1) a revised undergraduate measurements course in which the engineering concepts used to measure physical properties (2) revised undergraduate and graduate courses in which fluid flow are taught, and (3) a project in the capstone design course. Furthermore, master's thesis research of two minority students (Abel Vargas and Fernando Jasso) of Hispanic descend were supported through this grant. Abel Vargas completed his MS thesis in August 2002 and now attending the George Washington University to pursue his PhD degree. Fernando Jasso has finished his experiments and now writing his thesis. He is expected to defend his thesis in first week of July 2003.

Bibliography

Anderson, J. D. Jr. Computational Fluid Dynamics: The Basic Applications. New York: McGraw Hill, 1995.

Au, H. and Ko, N. W. M. "Coaxial Jets of Different Mean Velocity Ratios, Part 2." *Journal of Sound and Vibration*. 1987 Vol. 116 (3): 427-443.

Au, H. and Ko, N. W. M. "Coaxial Jets of Different Mean Velocity Ratios." *Journal of Sound and Vibration*. 1985. Vol. 100, No. 2: 211-232.

Basu, D. and Kumar, S. "Importing mesh entities through IGES/PDES", *Advances in Engineering Software*. 1995. Vol. 23: 151-161.

Beer, J. M., "Combustion Technology Developments In Power Generation In Response To Environmental Challenges." *Progress in Energy and Combustion Science*. 2000, Vol. 26: 301-327

Bennet, B. A. V., McEnally, C. S., Peefferle, L. D., Smooke, M. D., and Colket, M. B. "Computational and Experimental Study of Axisymmetric Coflow Partially Premixed Ethylene/Air Flames." *Combustion and Flame*. 2001. Vol. 127: 2004-2022.

Berger, S. A., Talbot L., and Yao, L.S. "Flow in Curved Pipes." *Annual Review of Fluid Mechanics*. 1983. Vol. 15: 461-512.

Britchford, K. M., Manner, A. P., McGuirk, J. J., and Stevens, S. J. "Measurements and Predictions of Flow in Annular S-Shaped Ducts." *Experimental Thermal and Fluid Science*. 1994. Vol. 9: 197-205.

Buresti, G., Petagna, P., and Tlamelli, A. "Experimental investigation on the turbulent near-field of coaxial jets." *Experimental Thermal and Fluid Sciences*. 1998. Vol. 17: 18-36.

CFD-ACETM Theory Manual Version 5. CFD Research Corporation, Huntsville, Alabama. 1998.

Champagne, F. H. and Wygnanski, I. J. "An Experimental Investigation of Coaxial Turbulent Jets." *International Journal of Heat and Mass Transfer*. 1971. Vol. 14: 1145-1464.

- Choudhuri, A.R., Luna, S. P., and Gollahalli, S. R. "Elliptical Coflow Effects on a Circular Gas Jet Flame." *Journal of Propulsion and Power*. 2002 Vol. 18, No. 3: 686-695.
- Crow, S. C., and Champagne, F. H. "Orderly structure in jet turbulence." *Journal of Fluid Mechanics*. 1971. Vol. 48: 547-591.
- Gollahalli, S. R., "Jet flames from noncircular burners." *Journal of Indian Academy of Sciences*. 1997. Vol. 22, No. 3: 369-382.
- Gutmark, E., and Ho, C. "Near Field Pressure Fluctuation of an Elliptic Jet." *AIAA Journal*. 1983. Vol. 23, No. 3: 354-358.
- Ho, C., and Gutmark, E. "Vortex induction and mass entrainment in a small-aspect-ratio elliptical jet." *Journal of Fluid Mechanics*. 1987. Vol. 179: 383-405.
- Huang, R. F., Yang, J., and Lee, P. "Flame and Flow Characteristics of Double Centric Jets." *Combustion and Flame*. 1997. Vol. 108: 9-23.
- Hussain, F. and Husain, H.S. "Elliptical jets. Part 1. Characteristics of unexcited and excited jet." *Journal of Fluid Mechanics*. 1989. Vol. 208: 257-320.
- Jain, R., and Jayaraman, G. "On the steady laminar flow in a curved pipe of varying elliptic cross-section." *Fluid Dynamic Research*. 1990. Vol. 5: 351-362.
- Ko N. W. M. and Lam, K. M. "Flow Structures of Coaxial Jet of Mean Velocity Ratio 0.5." *AIAA Journal*. 1989. Vol. 27, No. 5: 513-514.
- Lam, K. M. and Ko, N. W. M. "Investigation of Flow Structures of a Basic Annular Jet" *AIAA Journal*. 1986. Vol. 24, No. 9: 1488-1493.
- Lau, K. K., and Ko, N. W. M. "On the Formation of Wake Induced Structures in a Basic Annular Jet" 11th Australasian Fluid Mechanics Conference. Paper 4A-1. Dec.1992.
- Launder, B. E., and Spalding, D. B. "The Numerical Computation of Turbulent Flows." *Comput. Methods Applications Mech. Eng.* 1974. Vol. 3: 269-289.
- Malmstrom, T.G., Kirkpatrick, A.T., Christensen, B., and Knappmiller, K.D., "Centerline velocity decay measurements in low-velocity axisymmetric jets." *Journal of Fluid Mechanics*. 1997. Vol. 246: 363-377.
- Miller, R. S., Madnia, C. K., and Givi, P., "Numerical Simulation of Non-circular Jets." *Computers and Fluids*. 1995. Vol. 24: 1-25.

Montiel, E., Jasso, F., Castro, N. "Design and Installation of a Non-axisymmetric Coaxial Jet Test Facility." Senior Design Final Report, Department of Mechanical and Industrial Engineering, The University of Texas at El Paso. 2001.

Park, T. W., Katta, V. R., and Aggarwal, S. K., 1998, " On the Dynamics of A Two-Phase Non-Evaporating Swirling Jet," *International Journal of Multiphase Flow*, Vol. 24, pp. 295-317.

Patankar, S. V., Numerical Heat Transfer and Fluid Flow, New York: Hemisphere Publishing Company, 1980.

Quinn, W. R., "The Turbulent Free Jet Issuing from a Sharp-edged Elliptical Slot." AIAA Paper 89-0664 27th Aerospace Science Meeting. Reno, Nevada. 1989.

Ramesh, R.S., Pradeep, S., Muruganandam, T.M., and Sujith, R.I. "Studies on Free jets from Nozzles for High-speed Mixing Applications," *Experiments in Fluids*. 2000. Vol. 29: 359-368.

Ramos, J. I. "Drawing of annular liquid jets at low Reynolds number." *Computational and Theoretical Polymer Science*. 2001. Vol. 11: 429-443.

Raudkivi, A.J., and Callander, R.A. Advanced Fluid Mechanics An Introduction. New York: John Wiley & Sons Inc. 1975.

Savas, O. and Gollahalli, S. R., "Flow structure in the Near-Nozzle Region of Gas Jet Flames," *AIAA Journal*. 1986. Vol. 24: 1137-140.

Schadow, K. C., Gutmark, E., Koshigoe, S. and Wilson, K. J., "Combustion-Related Shear-Flow Dynamics in elliptic Supersonic Jets." *AIAA Journal*. 1989. Vol. 27 No. 10: 1347-1353.

Schadow, K. C., Gutmark, E., Parr, T. P., Parr, D. M., Wilson, K. J., and Ferrell, G. B. "Enhancement of Fine-Scale Mixing for Fuel Rich Plume Combustion" AIAA Paper 87-0376, Jan.1987.

Schadow, K. C., Wilson, K. J., and Lee M. J. "Enhancement of mixing inducted rockets with elliptic gas generator nozzles" AIAA Paper 84-1260, 20th Joint Propulsion Conference, Cincinnati, Ohio, Jan.1984.

Schlichting, H. "Laminare Strahlenausbreitung," *Z. Angew Math. Mech*. 1933. Vol. 13: 260-263.

Sforza, P. M., Steiger, M. H., and Trentacoste, N. "Studies on Three-Dimensional Viscous Jet." *AIAA Journal*. 1966. Vol. 4, No. 5: 800-806.

Sonntag, R. E., Borgnakke, C., and Van Wylen, G. J. Fundamentals of Thermodynamics. Fifth Edition. New York: John Wiley & Son, Inc. 1998.

Streeter, V. L., Wylie, E. B., Bedford, K. W., Fluid Mechanics. Ninth Edition. Boston: McGraw Hill 1998.

Tang, S. K. and Ko N. W. M. "Coherent Structure Interaction in an Excited Coaxial Jet of Mean Velocity Ratio of 0.3." *AIAA Journal*. 1993. Vol. 31, No. 8: 1521-1524.

Tang, S. K. and Ko N. W. M. "Experimental Investigation of the Structure Interaction in an Excited Coaxial Jet." *Experimental Thermal and Fluid Sciences*. 1997. Vol. 8: 214 – 229.

Tennekes, H., and Lumley, J. L. A First Course in Turbulence. Cambridge, Massachusetts: MIT Press 1972

Thomas, L. C. Heat Transfer. New Jersey: Prentice-Hall 1992.

Tritton, D. J., Physical Fluid Dynamics. Second Edition. Oxford: Clarendon Press 1988.

Turns, S. R. An Introduction to Combustion Concepts and Applications. New York: McGraw-Hill, 1996.

Versteeg H. K., and Malalasekera, W. An Introduction to Computational Fluid Dynamics: The Finite Volume Method. London: Prentice Hall, 1995.

Villiermaux, E., and Rehab, H., "Mixing in coaxial jets." *Journal of Fluid Mechanics*. 2000. Vol. 425: 161-185.

Wark, C., Eickmann, K., and Richards, C., 2000, "The Structure of An Acoustically Forced, Reacting Two-Phase Jet," *Combustion and Flame*, Vol. 120, pp. 539-548.

Warntz, J., Maas, U., and Dibble, R. W., Combustion Physical and Chemical Fundamentals, Modeling and Simulation, Experiment, Pollutant Formation. New York: Springer-Verlag. 1996.

White, F. Viscous Fluid Flow. 2nd Edition. Boston: McGraw Hill. 1991.

Wicker, R. B., Eaton, J.K., “Flow Visualization Studies of Structure of a Co-Flowing Jet with and without Controlled Excitation.” AIAA Paper 93-0651. 31st Aerospace Sciences Meeting & Exhibit. Reno, Nevada, 1993 Jan 11.

Wicker, R. B., Eaton, J. K. “Effect of Injected Longitudinal Vorticity on Particle Dispersion in a Swirling Coaxial Jet.” *Journal of Fluid Engineering*. 1999. Vol. 121, No. 4: 766-772.

www.tsi.com., 2002 TSI Incorporated.

Zaman, K. B. M. Q. “Axis switching and spreading of an asymmetric jet: the role of coherent structure dynamics.” *Journal of Fluid Dynamics*. 1996. Vol. 316: 1-27.

Appendix A – Derivation of Polynomial Profile Curves

Shown below are the general polynomial equations with letters $a - f$ being the coefficients to be solved; r is the radius and ℓ is the length of the curve.

- $r = a\ell^5 + b\ell^4 + c\ell^3 + d\ell^2 + e\ell + f$
- $r = b\ell^4 + c\ell^3 + d\ell^2 + e\ell + f$
- $r = c\ell^3 + d\ell^2 + e\ell + f$

Beginning the derivation of the 5th order polynomial equation

- Position Equation:

$$r = a\ell^5 + b\ell^4 + c\ell^3 + d\ell^2 + e\ell + f$$

- First Derivative: Slope

$$\frac{dr}{dx} = 5a\ell^4 + 4b\ell^3 + 3c\ell^2 + 2d\ell + e$$

- Second Derivative: Inflection Point

$$\frac{d^2r}{dx^2} = 20a\ell^3 + 12b\ell^2 + 6c\ell + 2d$$

Boundary Conditions

$x = 0$	$r = 3.799 \text{ inches}$	$\partial r / \partial x = 0$	$\partial^2 r / \partial x^2 = 0$
$x = \ell$	$r = 1.0 \text{ inch}$	$\partial r / \partial x = 0$	$\partial^2 r / \partial x^2 = 0$

Applying Boundary Conditions

$$\text{BC \#1: @ } x = 0 \quad r = 3.799 \text{ inches} \quad \partial r / \partial x = 0 \quad \partial^2 r / \partial x^2 = 0$$

$$r(0) = a\ell^5 + b\ell^4 + c\ell^3 + d\ell^2 + e\ell + f$$

$$3.799 = f$$

$$\frac{dr}{dx} = 5a\ell^4 + 4b\ell^3 + 3c\ell^2 + 2d\ell + e$$

$$0 = e$$

$$\frac{d^2r}{dx^2} = 20a\ell^3 + 12b\ell^2 + 6c\ell + 2d$$

$$0 = d$$

$$\text{BC \# 2: } x = \ell \quad r = 1.0 \text{ inch} \quad \partial r / \partial x = 0 \quad \partial^2 r / \partial x^2 = 0$$

$$r(\ell) = a\ell^5 + b\ell^4 + c\ell^3 + d\ell^2 + e\ell + f$$

$$1 = a\ell^5 + b\ell^4 + c\ell^3 + 3.799$$

$$-2.799 = a\ell^5 + b\ell^4 + c\ell^3$$

$$\frac{dr}{dx} = 5a\ell^4 + 4b\ell^3 + 3c\ell^2 + 2d\ell + e$$

$$0 = 5a\ell^4 + 4b\ell^3 + 3c\ell^2$$

$$\frac{d^2r}{dx^2} = 20a\ell^3 + 12b\ell^2 + 6c\ell + 2d$$

$$0 = 20a\ell^3 + 12b\ell^2 + 6c\ell$$

Simultaneous equations, three equations, three unknowns with $\ell = 4$ inches:

$$a(4)^5 + b(4)^4 + c(4)^3 = -2.799$$

$$5a(4)^4 + 4b(4)^3 + 3c(4)^2 = 0$$

$$20a(4)^3 + 12b(4)^2 + 6c(4) = 0$$

Coefficients after solving the simultaneous equations are:

$$a = -0.0164004$$

$$b = 0.1640039$$

$$c = -0.4373438$$

Substituting coefficients into the position equation yields:

$$r(\ell) = -0.0164004\ell^5 + 0.1640039\ell^4 - 0.4373438\ell^3 + 3.799$$

The 3rd and 4th order equations are generating by implementing the same method from above. For the curves of varying length, the variable ℓ is adjusted according. The outer wall profiles for the elliptical nozzle are also generated by employing the same method, although two curves on the minor and major axis plane must be generated. Each polynomial equation of the ellipse is distinct and are:

Major Axis $r_{maj} = -0.139736\ell^5 + 0.1397360\ell^4 - 0.372629339\ell^3 + 3.799$

Minor Axis $r_{min} = -0.1811668\ell^5 + 0.1811668\ell^4 - 0.483111467\ell^3 + 3.799$

Appendix B – Curve Importing Steps

1. Generate 200 x , y , and z coordinates in Microsoft Excel from the derived polynomial equation setting the z -component to zero.
2. Save the MS-Excel spreadsheet as a Tab delimited file (.txt)
3. Change the Tab Delimited file to an ASCII file. This is done by changing the extension of the text file by right clicking on file, go to **Properties**, and then change **.txt** to **.pt**
4. Open Thinkdesign™ 6.0 software.

- Click on Fit Curve icon

- Select Import points from ASCII file

- Find the file name with extension **.pt**, and double click.
- In the **Point fitting data**, click on constrained and then **OK**.

- Select **Both**

- In the Start tangent menu, select **Enter the components**. Enter the values of the derivatives as shown below. Click **OK** when finished.

- In the **End tangent** menu, select **Enter the components**. Enter the values of the derivatives as shown below. Click **OK** when finished.

- Keep the default values as shown below for the **Curve Data** menu and click **OK**.

- Cancel the **Fitting** menu, and the curve appears on the screen.

5. Save the file as an IGES file.
6. Open CFD-GEOM and import the IGES file.

Appendix C – Hot Wire Probe Calibration Procedures

The following were the steps necessary to calibrate the tungsten hot wire probe. Proper calibration of the hot wire ensures accurate and reliable velocity measurement. All instrumentation was connected as show in the schematic below.

Calibration Procedures

1. The pressure transducer output is connected to the pressure differential (dP) input on the back panel of the IFA 300 Constant Temperature Anemometer.
2. The pressure transducer resistance was calibrated by adjusting the internal screw to stabilize the resistance to zero ohms. A digital multi-meter was used to verify the resistance.
3. The thermocouple was placed in the calibrator and connected to the thermocouple connector of the IFA 300.
4. The arm of the calibrator was placed on the horizontal position that is marked with an H at the side of the calibrator.
5. The probe support was placed and secured.
6. The ThermalPRO software was opened and the **Probe Data** option was selected
7. **s_film.cl** was selected from the list of the data files.
8. The shorting probe placed into the support probe and the **Read** button selected. The value output is typically less than 0.5 ohms.

9. The appropriate values for the following parameters are entered:

Probe Type	(S) for single probe
Wire film	(W) for wire
Offset and Gain	1.8 and 3
Cal Method	Acquire E & Acquire dP

10. **Autocal Tbl** button was selected. A list of calibration tables appears. The table with filename **M140.ctb**. was selected and saved with a new profile name, preferably the serial number of the hot-wire sensor.

11. Probe placed in probe support. The probe is in the vertical position “V”, just directly above the calibrator nozzle.

12. The parameter for the **Condition Setup** screen appears. This screen provides the calibration parameter, which included: temperature, minimum velocity, maximum velocity, and atmospheric pressure.

13. The software was ready to acquire calibration points. The valves were closed before acquiring calibration points to prevent air flow into the calibrator unit.

14. Displayed values were observed in the **Current dP** box, when it read zero **Acquire** was selected.

15. The calibration software automatically indicates the next calibration point and a new value for the next dP is displayed.

16. The calibration flow adjusted by opening and closing the coarse and fine flow valves to obtain a reference starting point.