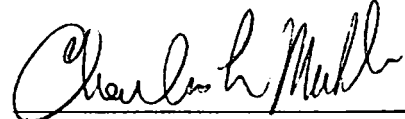


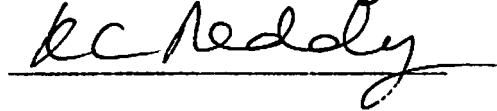
To the Graduate Council:

I am submitting herewith a dissertation written by Charles Lynn Sebourn entitled "Computational Investigation of the Discharge Coefficient of Bellmouth Flow Meters in Engine Test Facilities." I have examined the final paper copy of this dissertation for form and content and recommend that it be accepted in partial fulfillment of the requirements for the degree of Doctor of Philosophy, with a major in Aerospace Engineering.

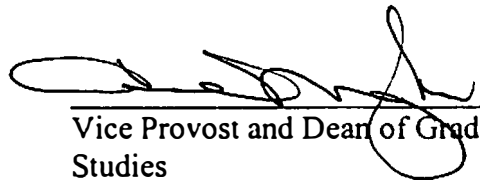


Dr. Charles Merkle, Major Professor

We have read this dissertation
and recommend its acceptance:



Accepted for the Council:



Vice Provost and Dean of Graduate
Studies

Handwritten text, possibly a date or reference number, located at the top center of the page.

Handwritten text, possibly a signature or name, located in the middle right section of the page.

Handwritten text, possibly a signature or name, located in the bottom left section of the page.

**COMPUTATIONAL INVESTIGATION OF THE DISCHARGE COEFFICIENT OF
BELLMOUTH FLOW METERS IN ENGINE TEST FACILITIES**

**A Dissertation
Presented for the
Doctor of Philosophy
Degree
The University of Tennessee, Knoxville**

**Charles Lynn Sebourn
May, 2002**

Thesis
2002b
.S43

DEDICATION

This dissertation is dedicated to my wife Teresa and two children, Lyndsey and Jared. It is their sacrifice and support that has made this effort possible.

ACKNOWLEDGMENTS

I would like to acknowledge the United States Air Force and Sverdrup Technology, Inc. for the support provided at the Arnold Engineering Development Center (AEDC) and during my year of off-site study as recipient of the Resident Graduate Study Program scholarship. Thanks is also due the many individuals who have supported and encouraged this course of study.

I would like to thank Barry Bishop who first encouraged me to pursue this course of study as well as Tom McGill and Joe Hollis for supporting my candidacy to the Resident Graduate Study Program.

Thanks is due to my work team members, who have had to tolerate my studies while working with me. These include Lou Deken, Rick Burdette, Glen Merz, Danny Sells, and Russ Groff, who will be pleased to know that he has avoided the “and others” category.

Special thanks to Brent Bates who has worked closely with me allowing me to bounce ideas and complaints off of him and serving as a peer reviewer for this paper.

Thanks to Dave Beale, whose airflow measurement project provided the critical data for this study and whose investigations were largely responsible for prompting this research. Dave Beale and Terry Hand both provided critical feedback on this body of research. Thanks also to Jim Sirbaugh for providing peer review and assistance with the rotating frame. Thanks to Dr. Greg Power and Chris Nelson for assisting with the WIND code.

Thanks go to the management of the Applied Technology who provided needed encouragement and prodding. These include Dr. Ralph R. Jones, III, Dr. Tim Swafford, and Bill Muse.

Thanks go to all the committee members especially Dr. Merkle, who was a joy to work with and is especially to be commended for the professional respect with which he treats all of his students.

Special thanks also to Dr. Virgil K. Smith, III for providing oversight of the Resident Graduate Study Program and for his excellent advice on navigating the pitfalls and finer points of the approval process.

The work reported herein was conducted at the Arnold Engineering Development Center (AEDC), Air Force Material Command (AFMC). The results were obtained by Sverdrup Technology, Inc., AEDC Group, operating contractor for test facilities at AEDC, AFMC, Arnold Air Force Base, Tennessee under Job 1029.

Finally, a special word of thanks to those who have provided support at home through difficult family health issues during the course of this research. These include the members of the Westwood Church of Christ in Tullahoma, Tennessee and my in-laws, Jeff and Cindy Scott, and Brent and Melissa Green. Finally, a special thanks to my wife's parents, Ray and Shirley Wilmoth and my own parents, Charles and Anna Mary Sebourn.

ABSTRACT

In this thesis computation of the discharge coefficient of bellmouth flow meters installed in engine test facilities is presented. The discharge coefficient is a critical parameter for accurately calculating flow rate in any flow meter which operates by means of creating a pressure differential. Engine airflow is a critical performance parameter and therefore, it is necessary for engine test facilities to accurately measure airflow.

In this report the author investigates the use of computational fluid dynamics using finite difference methods to calculate the flow in bellmouth flow meters and hence the discharge coefficient at any measurement station desired.

Experimental boundary layer and core flow data was used to verify the capability of the WIND code to calculate the discharge coefficient accurately. Good results were obtained for Reynolds numbers equal to or greater than about three million which is the primary range of interest.

After verifying the WIND code performance, results were calculated for a range of Reynolds numbers and Mach numbers. Also the variation in discharge coefficient as a function of measurement location was examined. It is demonstrated that by picking the proper location for pressure measurement, sensitivity to measurement location can be minimized.

Also of interest was the effect of bellmouth geometry. Calculations were performed to investigate the effect of duct to bellmouth diameter ratio and the eccentricity of the bellmouth contraction. In general the effects of the beta ratio were seen

to be quite small. For the eccentricity, the variation in discharge coefficient was as high as several percent for axial locations less than half a diameter downstream from the throat.

The second portion of the thesis examined the effect of a turbofan engine stationed just downstream of the bellmouth flow meter. The study approximated this effect by examining a single fan stage installed in the duct. This calculation was performed by making use of a rotating frame of reference.

Initial attempts to perform such a calculation were unsuccessful. Careful examination of the results revealed deficiencies in the rotating frame capabilities of the WIND code. These deficiencies were corrected and a test case was performed to demonstrate the correct calculation of the rotating frame.

A well known NASA test article, Rotor 67, was selected to emulate the presence of a turbofan engine in the bellmouth duct. Results indicated a very complex flow field upstream of the engine face. The primary disturbance of the flow was the propagation of a shock off the fan blade leading edge. The rotation of the blades traces out a helical pressure distortion which travels upstream. Interference patterns also develop which add complexity to the wall static pressure. Induced rotation of the flow upstream of the fan was found to be very small.

The pressure fluctuations on the wall are periodic and varied with location. However, when the results are time averaged the resulting pressure and hence discharge coefficient curve is seen to be smooth and well-behaved.

As a result of this study the application of computation fluid dynamic techniques has been verified for the calculation of discharge coefficient in bellmouth flow meters. The results also provide insight on measurement location and technique. As a result of the fan calculation, the WIND code has been improved and a capability to calculate rotating flows in a duct has been developed. Finally, results in the rotating frame have provided insight on discharge coefficient behavior in the presence of a rotating engine component.

Table of Contents

SECTION	Page
1. Introduction.....	1
1.1. Background.....	3
1.1.1. General Operation of a Bellmouth Flow Meter.....	3
1.1.2. Calculation of Discharge Coefficient	4
1.2. AEDC Bellmouth Airflow Measurement	8
1.3. Statement of Problem	10
2. Literature Survey	12
2.1. ASME Long-Radius Flow Nozzle.....	12
2.1.1. Nozzle Description	12
2.1.2. Discharge Coefficient.....	13
2.2. Review of Incompressible Flow Meter Literature.....	14
2.3. Performance of Flow Meters in Compressible Flow	25
3. Calculation of Bellmouth Discharge Coefficient.....	30
3.1. Description of Engine Facility Bellmouth.....	30
3.2. WIND CODE Description.....	31
3.3. Description of Bellmouth Test and Data Set	33
3.3.1. Examination of Data.....	35
3.4. WIND Computation of Bellmouth Flow	37
3.4.1. Description of Grid.....	37
3.4.2. Turbulence Model	40

3.4.3. Typical Results	43
3.4.4. Comparison with Experiment.....	49
3.4.5. Discussion of Computational Results.....	50
3.4.6. Comparison to ASME Flow Nozzle.....	53
3.4.7. Effects of Bellmouth Inlet Contraction Eccentricity	55
3.4.8. Effects of Beta Ratio	56
4. 3-D Effect of Rotating Fan Installed in Bellmouth Flow Meter.....	57
4.1. Engine Proximity to Bellmouth Flow Meter	57
4.2. Approach.....	57
4.2.1. Navier-Stokes Equations in a Rotating Frame of Reference.....	58
4.2.2. WIND Code Advertised Capabilities	62
4.3. Rotating Frame Test Case.....	63
4.4. Corrections to the WIND Code Rotating Frame of Reference Capability	65
4.4.1. Inlet Boundary Total Pressure and Total Temperature	65
4.4.2. Inlet Characteristics for Rotational Frame	67
4.4.3. Wall Boundary Pressure	68
4.4.4. Wall Velocity for Slip Wall.....	68
4.4.5. Coupled Boundary.....	69
4.4.6. Singular Axis Corrections	71
4.4.7. Corrections Involving Body Forces.....	72
4.4.8. Corrections involving the Roe Method	75
4.4.9. Turbulence Model Corrections.....	78

4.4.10. Speed of Sound Corrections:	78
4.5. Demonstration of Rotating Frame Corrections.....	79
4.6. Fan Selection	82
4.7. Computational Grids.....	83
4.8. Computational Results of Fan Installed in Straight Duct	85
4.8.1. Comparison of Results with Experiment.....	87
4.8.2. Examination of Results in a Straight Duct	89
4.8.3. Discharge Coefficient in Straight Duct Upstream of Fan	93
4.9. Computational Results for Bellmouth Upstream of the Fan	96
4.9.1. Discharge Coefficient of Bellmouth with Installed Fan.....	97
5. Summary and Conclusions	101
6. Recommendations for Future Work.....	106
Bibliography	110
Appendices.....	115
A. Tables	116
B. Figures.....	118
Vita.....	223

List of Tables

TABLE	PAGE
1 Mach and Reynolds Number Conditions for the WIND-based CFD Simulations of Bellmouth Flows	117

List of Figures

FIGURE	PAGE
1 Typical Turbine Engine Test Cell Showing Location of Flow Measuring Venturis.	119
2 ASME Flow Meter Recommended Geometry.	119
3 Location of Pressure Taps Used With ASME Flow Nozzles.....	120
4 ASME Flow Nozzle Coefficients.....	121
5 ASME Flow Nozzle Discharge Coefficient as a Function of Beta Ratio.....	122
6 Bellmouth Experiment Schematic.....	123
7 Bellmouth Contraction Geometry.	123
8 Installed Experimental Bellmouth Configuration.	124
9 Boundary Layer Rake Detail	124
10 Surface Measurement Detail.	125
11 Experimental Boundary Layer Total Pressure Probe Data, Station 3, Mach = 0.31.	126
12 Experimental Boundary Layer Total Pressure Probe Data, Station 3, Mach = 0.4.	126
13 Experimental Boundary Layer Total Pressure Probe Data, Station 3, Mach = 0.5.	127
14 Experimental Boundary Layer Total Pressure Probe Data, Station 3, Mach = 0.7.	127

15	Experimental Boundary Layer Total Pressure Probe Data, Station 8, Mach = 0.31.	128
16	Experimental Boundary Layer Total Pressure Probe Data, Station 8, Mach = 0.4.	128
17	Experimental Boundary Layer Total Pressure Probe Data, Station 8, Mach = 0.5.	129
18	Experimental Boundary Layer Total Pressure Probe Data, Station 8, Mach = 0.7.	129
19	Experimental Displacement Thickness at Station 3 for Various Reynolds No.	130
20	Experimental Displacement Thickness at Station 8 for Various Reynolds No.	130
21	Discharge Coefficient at Station 3 Based on Displacement Thickness.	131
22	Discharge Coefficient at Station 8 Based on Displacement Thickness.	131
23	Bellmouth Computational Grid.	132
24	Computed Wall Static Pressure at Mach = 0.5 and Reynolds No. = 5.75E6 for Various Grid Resolutions.	132
25	Computed Discharge Coefficient in the Throat Region, Mach = 0.5, Reynolds No. = 5.65E6 for Various Grid Resolutions.	133
26	Computed Boundary Layer Mass Flux at Station 8, M=0.5, Reynolds No. = 5.75E6 for Various Grid Resolutions.	133
27	Computed Mass Flux at Station 8, Mach =0.5, Reynolds No. = 5.75E6 for Various Grid Resolutions.	134

28	Discharge Coefficient at Station 3 Computed Using Various Grid Resolutions, Mach = 0.5.....	134
29	Discharge Coefficient at Station 8 Computed Using Various Grid Resolutions, Mach = 0.5.....	135
30	Discharge Coefficient at Station 3 Computed Using Various Grid Resolutions, Re = 5.75E6.....	135
31	Discharge Coefficient at Station 8 Computed Using Various Grid Resolutions, Re = 5.75E6.....	136
32	Comparison of Turbulence Model Result for Total Pressure Profiles to Experimental Data at Station 3, Mach = 0.5, Re = 2E6.	136
33	Comparison of Turbulence Model Results for Total Pressure Profile to Experimental Data at Station 3, Mach = 0.5, Re = 3E6.	137
34	Comparison of Turbulence Model Results for Total Pressure Profile to Experimental Data at Station 3, Mach = 0.5, Re = 3.9E6.	137
35	Comparison of Turbulence Model Results for Total Pressure Profile to Experimental Data at Station 3, Mach = 0.5, Re = 5.75E6.	138
36	Comparison of Turbulence Model Results for Total Pressure Profile to Experimental Data at Station 8, Mach=0.5, Re = 2E6.	138
37	Comparison of Turbulence Model Results for Total Pressure Profiles to Experimental Data at Station 8, Mach = 0.5, Re = 3E6.	139
38	Comparison of Turbulence Model Results for Total Pressure Profiles to Experimental Data at Station 8, Mach = 0.5, Re = 3.9E6.	139

39	Comparison of Turbulence Model Results for Total Pressure Profiles to Experimental Data at Station 8, Mach 0.5, $Re = 5.75E6$	140
40	Comparison Between Experimental and Computed Displacement Thickness for Various Turbulence Models at Station 3.....	140
41	Comparison of Experimental and Computed Momentum Thickness for Various Turbulence Models at Station 3.....	141
42	Comparison of Experimental and Computed Displacement Thickness for Various Turbulence Models at Station 8.....	141
43	Comparison of Experimental and Computed Momentum Thickness for Various Turbulence Models at Station 8.....	142
44	Comparison of Calculated Discharge Coefficient at Stations 8 for Various Turbulence Models.....	142
45	Comparison of Turbulence Model Results for Total Pressure Profiles at Half Nominal Grid Resolution: Station 3, $M=0.5$, $Re=3.9E6$	143
46	Comparison of Turbulence Model Results for Total Pressure Profiles at Half Nominal Grid Resolution: Station 8, $M=0.5$, $Re=3.9E6$	143
47	Comparison of Turbulence Model Results for Total Pressure Profiles at Double Nominal Grid Resolution: Station 3, $M=0.5$, $Re=3.9E6$	144
48	Comparison of Turbulence Model Results for Total Pressure Profiles at Double Nominal Grid Resolution: Station 8, $M=0.5$, $Re=3.9E6$	144
49	Max Residual, $M=0.5$, $Re=5.75E6$	145
50	L2-norm Residual, $M=0.5$, $Re=5.75E6$	145

51	Calculated Mass Flow, $M=0.5$, $Re=5.75E6$	146
52	Pressure Contour, $Mach = 0.5$, $Re = 5.75E6$	147
53	Mach Contour, $Mach = 0.5$, $Re = 5.75E6$	147
54	Velocity Contour, $Mach = 0.5$, $Re = 5.75E6$	148
55	Mass Flux Contours, $Mach =0.5$, $Re = 5.75E6$	148
56	Wall Static Pressure, $M=0.5$, $Re=5.75E6$	149
57	Discharge Coefficient and Wall Profile, $M=0.5$, $Re=5.75E6$	149
58	Discharge Coefficient in the Throat Region, $Mach = 0.5$, $Re = 5.75E6$	150
59	Discharge Coefficient Slope, $M=0.5$, $Re=5.75E6$	150
60	Comparison of Predicted and Experimental Total Pressure Ratio in Boundary Layer, $M=0.5$, $Re=5.75E6$	151
61	Mass Flux in Boundary Layer, $M=0.5$, $Re=5.75$	151
62	Mass Flux, $Mach = 0.5$, $Re=5.75E6$	152
63	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.31$, $Re=2E6$	152
64	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.31$, $Re=3E6$	153
65	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.31$, $Re=3.9E6$	153
66	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.31$, $Re=5.75E6$	154

67	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.4$, $Re=2E6$	154
68	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.4$, $Re=3E6$	155
69	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M = 0.4$, $Re=3.9E6$	155
70	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.4$, $Re=5.75E6$	156
71	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.5$, $Re=2E6$	156
72	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.5$, $Re=3E6$	157
73	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.5$, $Re=3.9E6$	157
74	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.5$, $Re=5.75E6$	158
75	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.7$, $Re=2E6$	158
76	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.7$, $Re=3E6$	159
77	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.7$, $Re=3.9E6$	159

78	Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.7$, $Re=5.75E6$	160
79	Discharge Coefficient Variation with Reynolds No. at Station 3.	160
80	Discharge Coefficient Variation with Reynolds No. at Station 8.	161
81	Discharge Coefficient Variation with Mach Number at Station 3.	161
82	Discharge Coefficient Variation with Mach Number at Station 8.	162
83	Axial Variation of Discharge Coefficient for Various Mach Numbers, Reynolds No. = $5.75E6$	162
84	Comparison of Predicted Mass Flux at Station 8, Reynolds No. = $5.75E6$	163
85	Comparison of Mass Flux Contours, $Re = 2E6$	164
86	Calibration Curve of a Low Beta Flow Nozzle with Throat Pressure Taps.	165
87	Static Pressure Tap Error as a Function of Wall Shear Stress and Hole Diameter.	165
88	Comparison of Computed Results at Mach 0.2 with ASME Recommended Values.	166
89	Range of Elliptical Inlet Profiles for Bellmouth (a – major axis, b – minor axis). ·	166
90	Axial Distribution of the Computed Discharge Coefficient for Various Axis Ratios, $M=0.5$, $Re = 3E6$	167
91	Axial Distribution of the Computed Discharge Coefficient for Various Axis Ratios in the Throat Region, $M = 0.5$, $Re = 3E6$	167
92	Axial Distribution of the Percent Variation of the Computed Discharge Coefficient Between $b/a = 0.3$ and $b/a = 1.0$; $M=0.5$, $Re = 3E6$	168

93	Variation of the Computed Discharge Coefficient as a Function of Axis Ratio, M=0.5, Re=3E6.	168
94	Axial Distribution of the Computed Discharge Coefficient for Various Beta Ratios, M = 0.5, Re = 3E6.	169
95	Axial Distribution of the Computed Discharge Coefficient for Various Beta Ratios in Throat Region, M = 0.5, Re = 3E6.	169
96	Variation of Discharge Coefficient with Beta Ratio.	170
97	Comparison of Reference Frames.	170
98	Cylinder Test Case Grid.	171
99	Cylinder Test Case Grid: End View.	171
100	Mach Contours for Rotating Frame Test Case.	172
101	Mach Contours for Inertial Frame Test Case.	172
102	Comparison of Predicted Mach Number Along Vertical Radius.	173
103	Comparison of Predicted Mach Number Along Circumferential j=25 Grid Line.	173
104	Predicted Pressure Contours for Rotating Frame Test Case.	174
105	Predicted Pressure Contours for Inertial Frame Test Case.	174
106	Comparison of Predicted Pressure Along Vertical Radius.	175
107	Comparison of Predicted Pressure Along j=25 Grid Line.	175
108	Flow Vectors in the Y-Z Plane for Rotating Frame Test Case.	176
109	Velocity Vector in Rotating Frame At Uncorrected Slip Wall Boundary.	177
110	Velocity Vector in Inertial Frame At Uncorrected Slip Wall Boundary.	177
111	Velocity Vectors in Rotating Frame At Corrected Slip Wall Boundary.	178

112	Velocity Vectors in Inertial Frame at Corrected Slip Wall Boundary.	178
113	Cylindrical Test Grid With Coupled Boundary Highlighted.	179
114	Velocity Vectors in Rotating Frame for Cylindrical Test Case.	179
115	Velocity Vectors in Inertial Frame at Uncorrected Coupled Boundary.	180
116	Illustration of Coupling Error for Z Velocity Component on Vertical Coupled Boundary.	180
117	Illustration of Coupling Error for Y Velocity Component on Vertical Coupled Boundary.	181
118	Detail of Coupling Error for Y Velocity Component on Vertical Coupled Boundary.	181
119	Comparison of Z Component of Velocity for Corrected and Uncorrected Boundary Layer Coupling.	182
120	Singular Axis Averaging and Relaxation Schematic.	182
121	Y Velocity Component After Single Iteration.....	183
122	Z Velocity Component After A Single Iteration.	183
123	WIND Code Control Volumes.	184
124	Roe Method Comparison of Inertial and Rotating Frame.	185
125	Comparison of Predicted Mach Number Along Vertical Radius of Simple Cylinder for Corrected and Uncorrected Rotation.	186
126	Comparison of Predicted Mach Number Along j=25 Grid Line for Corrected and Uncorrected Rotation.	186

127	Comparison of Predicted Pressure Along Vertical Radius of Simple Cylinder for Corrected and Uncorrected Rotation.	187
128	Comparison of Predicted Pressure Along Vertical Radius of Simple Cylinder for Corrected and Uncorrected Rotation.	187
129	Three Dimensional Bellmouth Grid.	188
130	Grid in Single k-Plane for Comparison Results.	188
131	Comparison of Predicted Centerline Velocity for Corrected Rotating and Nonrotating Solution.	189
132	Comparison of Predicted Centerline Pressure for Corrected Rotating and Nonrotating Solution.	189
133	Comparison of Predicted Wall Pressure for Corrected Rotating and Nonrotating Solution.....	190
134	Comparison of Predicted Axial Component of Velocity at Station 8 for Corrected Rotating and Nonrotating Solution.....	190
135	Comparison of Predicted Y Component of Velocity at Station 8 for Corrected Rotating and Nonrotating Solution.....	191
136	Comparison of Predicted Z Component of Velocity at Station 8 for Corrected Rotating and Nonrotating Solution.....	191
137	Comparison of Predicted Pressure at Station 8 for Corrected Rotating and Nonrotating Solution.	192
138	Comparison of Predicted Temperature at Station 8 for Corrected Rotating and Nonrotating Solution.	192

139	Predicted Pressure Along the $j=25$ Gridline for Corrected Rotating and Nonrotating Solution.	193
140	Predicted Temperature Along the $j=25$ Gridline for Corrected Rotating and Nonrotating Solution.	193
141	Rotor 67.	194
142	Rotor 67 Pressure Ratio Performance.	195
143	Contour Plots of Rotor 67 Relative Mach Number.	196
144	Model of Rotor 67 with Spherical Nose.	197
145	Side View of Rotor 67 Model.	197
146	Front View of Rotor 67 Model.	198
147	Grid of Fan with Straight Inlet Duct.	198
148	Close Up of Grid in the Region of the Fan.	199
149	Side View of Grid: Fan with Straight Duct.	199
150	Top View of Grid Around Fan Blade.	200
151	Grid Computational Zones Fan in a Straight Duct.	200
152	Side View of Grid: Fan with Bellmouth Inlet.	201
153	Computational Zones for Fan with Bellmouth Inlet.	201
154	Computed Mach Contours At 10% of Span From Tip.	202
155	Computed Mach Contours At 30% of Span From Tip.	202
156	Computed Mach Contours At 70% Of Span From Tip.	203
157	Comparison of Experimental and Computed Total Pressure At Aerodynamic Station 1.	203

158	Comparison of Experimental and Computed Total Temperature at Aerodynamic Station 1	204
159	Comparison of Experimental and Computed Static Pressure at Aerodynamic Station 1	204
160	Predicted Contours of Mach Number in a Constant Area Inlet Duct	205
161	Predicted Contours of X-Component of Velocity in a Constant Area Inlet Duct ..	205
162	Predicted Contours of Y-Component of Velocity in a Constant Area Inlet Duct. .	206
163	Predicted Contours of Z-Component of Velocity in a Constant Area Inlet Duct. .	206
164	Predicted Contours of Static Temperature Contours in a Constant Area Inlet Duct.	207
165	Predicted Contours of Static Pressure in a Constant Area Inlet Duct.	207
166	Predicted Static Pressure Field at Wall in the Region of the Fan.....	208
167	Detail of Shock at Fan Blade Tips.....	208
168	Three-Dimensional View of Static Pressure on Duct Wall.....	209
169	Detail of Static Pressure on Duct Wall.....	209
170	Predicted Pressure Contours at $x = -0.2$	210
171	Predicted Pressure Contours at $x = -0.5$	210
172	Predicted Pressure Contours at $x = -1$	211
173	Predicted Pressure Contours at $x = -1.5$	211
174	Predicted Pressure Contours at $x = -2$	212
175	Predicted Pressure Contours at $x = -2.5$	212
176	Predicted Pressure Contours at $x = -3$	213

177	Predicted Pressure Contours at $x = -3.5$	213
178	Predicted Pressure Contours at $x = -4$	214
179	Instantaneous Static Pressure On Wall vs. X-Coordinate.	214
180	Predicted Distributions of Instantaneous Static Pressure On Wall At Constant X Plane.	215
181	Max, Min, and Average Static Pressure on Wall vs. X-Coordinate.....	215
182	Computed Discharge Coefficient Along Duct Based on Average Static Pressure.....	216
183	Predicted Mach Number Contours in Bellmouth Upstream of Fan.	216
184	Predicted X-Component of Velocity Contours in Bellmouth Upstream of Fan. ...	217
185	Predicted Y-Component of Velocity Contours in Bellmouth Upstream of Fan. ...	217
186	Predicted Z-Component of Velocity Contours in Bellmouth Upstream of Fan.	218
187	Predicted Temperature Contours in Bellmouth Upstream of Fan.	218
188	Predicted Static Pressure Contours in Bellmouth Upstream of Fan.	219
189	Predicted Static Pressure Contours in Bellmouth Throat Region.	219
190	Predicted Static Pressure Contours at the Wall in Throat Region of Bellmouth....	220
191	Instantaneous Wall Pressure Along Axis of Bellmouth.	220
192	Wall Static Pressure at Constant Axial Planes in Throat Region.	221
193	Maximum, Minimum, and Average Wall Pressure Along Axis of Bellmouth.	221
194	Computed Discharge Coefficient Along Axis of Bellmouth with Fan.	222
195	Comparison of Computed Discharge Coefficient in Bellmouth With Fan and Without Fan.	222

LIST OF SYMBOLS

Symbol	Definition
A	Flow area
c	Speed of sound
c_v	Specific heat at constant volume
C_D	Discharge coefficient
C_3	Third flow characteristic
d	Bellmouth flow meter throat diameter
e	Internal energy
E	Total energy in the inertial frame
E^*	$H - \tilde{u}\tilde{v}$, “rotational energy”
D	Upstream pipe diameter
$\bar{f}_{APP}$	Apparent friction coefficient
$\tilde{f}_e$	Body force vector
$\tilde{F}_{centrifugal}$	Centrifugal force
$\tilde{F}_{coriolis}$	Coriolis force
h	Enthalpy
H	Total enthalpy in the inertial frame
i	First computational coordinate
I	Rothalpy

$\bar{\mathbf{I}}$	Identity matrix
j	Second computational coordinate
k	Third computational coordinate
k	Thermal conductivity
$\dot{m}$	Mass flow rate
$\dot{m}_{actual}$	Actual mass flow rate through flow meter
$\dot{m}_{ideal}$	Ideal mass flow rate through flow meter
M	Mach number
M_{cv}	Mass of control volume
M_{ideal}	Ideal Mach number based on apparent pressure ratio
M_I	Mach number in the inertial frame of reference
M_R	Mach number in the rotating frame of reference
p	Static pressure
$p_{measured}$	Measured static pressure
p_T	Total pressure
p_0	Local Total Pressure
$p_{0_{int}}$	Plenum Inlet Total Pressure
q_H	Energy of heating per unit volume
$\tilde{q}$	Flow solution vector
$\tilde{q}_L$	Left cell boundary state

$\tilde{q}_R$	Right cell boundary state
r	Pressure ratio, radius
r_{wall}	Radius at wall
$\tilde{\mathbf{r}}$	Radius vector
$\bar{r}_x$	x -coordinate of control volume center of gravity
$\tilde{\mathbf{r}}$	Control volume center of mass
R	Ideal gas constant
Re_D	Reynolds number based on pipe diameter
t	Time
T	Static temperature
T_t	Total temperature
u_x	x component of rotating frame velocity
u_y	y component of rotating frame velocity
u_z	z component of rotating frame velocity
$\tilde{\mathbf{u}}$	Velocity of the rotating frame
$\tilde{\mathbf{u}}_{centroid}$	Rotating frame velocity at the control volume center of mass
U	Fluid velocity at bellmouth exit
U_n	Velocity component normal to inlet boundary
U_p	Velocity component parallel to inlet boundary
v_x	x component of velocity in the inertial frame of reference

v_y	y component of velocity in the inertial frame of reference
v_z	z component of velocity in the inertial frame of reference
$\vec{v}$	Velocity vector in the inertial frame of reference
V	Volume
w_f	Fluid work per unit volume
w_x	x component of velocity in the rotating frame
w_y	y component of velocity in the rotating frame
w_z	z component of velocity in the rotating frame
$\vec{w}$	Velocity vector in the rotating frame of reference
x	Distance along the first Cartesian coordinate
y	Distance along the second Cartesian coordinate
Y	Compressibility correction factor
z	Distance along the third Cartesian coordinate
β	Flow meter diameter ratio, d / D
δ^*	Displacement thickness
γ	Ratio of specific heats
ρ	Density
$\bar{\bar{\tau}}$	Stress tensor
$\vec{\omega}$	Rate of rotation of rotating frame of reference
ω_x	Rate of rotation about the x-axis
ω_y	Rate of rotation about the y-axis

ω_z Rate of rotation about the z-axis

Ω Vorticity

1. Introduction

Turbine engine ground test facilities provide the aircraft systems developer the means to evaluate turbine engine performance, operability, and durability. Performance testing includes such parameters as turbine engine thrust and fuel consumption. Operability testing involves parameters such as engine surge margin when subjected to distorted inlet flow. Durability testing requires evaluations of the structural integrity of various system components. However, for most forms of testing, the ground test facilities must provide some common services and information. The subject of this document will be the process to quantify the engine airflow rate accurately.

The Arnold Engineering Development Center (AEDC) offers a number of direct-connect turbine engine test cells for performance testing of turbine engines. These tests emphasize airflow measurement accuracy. This is a result of the influence of airflow rate on performance parameters, especially fuel consumption, which is a prime consideration of engine performance. As a result, AEDC commercial engine customers have requested airflow measurement accuracy goals as low as 0.23%. Extended range has also become an important criteria in military aircraft performance, increasing the importance of airflow measurement for military engines.

The primary method at the AEDC facilities for measuring airflow rates is through the use of critical flow venturis. A typical installation in a turbine engine cell illustrating flow measurement components is shown in Figure 1. (Figures are shown in Appendix B.) The venturi consists of a choked converging-diverging nozzle. The airflow rate is derived

from measurements of total pressure, total temperature, and the venturi throat area. The performance of the critical flow venturi is well established [1]. The critical flow venturi is capable of providing accurate airflow measurements, typically 0.4%, however there are a number of disadvantages in venturi operation.

First, the critical flow operation results in a substantial loss of total pressure as the flow passes through the venturi. The pressure loss is a result of shock waves in the diverging section and turbulence that is generated by the shocks. The resulting pressure loss must be compensated for by increased supply pressure delivered by the facility air compressors at a corresponding higher utility cost.

Second, provisions must be made to provide for variation in airflow rates, to accommodate multiple test conditions. This requires adjusting the effective venturi throat area to maintain choked operation. This is accomplished by either physically exchanging venturis to match throat diameter with airflow demands, or by operating an array of smaller venturis that can be individually opened and closed to provide the proper total flow rate. Physical exchange of the venturi requires a disassembly process which must be generally executed between test periods and involves significant labor charges. Operation of an array of venturis requires test time for operation as well as costs associated with venturi maintenance.

Finally, operation with choked venturis leads to a slow response in changing airflow rates. A change in airflow rate is accomplished only through a change in the supply pressure. Rapid changes in airflow, while remaining in the choked regime, are difficult to attain. Therefore, the venturis usually must be removed to accommodate

testing of engine transient operations. This installation and removal process also introduces time and monetary costs into turbine engine test programs.

A program was carried out to acquire accurate airflow measurements at AEDC while reducing the inherent costs associated with the venturi. The approach consisted of developing a technique for using a bellmouth flow meter as an alternative to the venturi flow meter. The bellmouth serves as a transitional section between the upstream plenum chamber and the engine air supply duct. A bellmouth of some configuration is present in nearly all engine tests and is located close to the test article. It is not limited by the flow and operability restrictions of the venturi noted above. Furthermore, the bellmouth operates subcritically and therefore does not generate a much smaller total pressure loss in comparison to the venturis. Therefore, if proper accuracy can be established for the wide variety of bellmouths used, the bellmouth flow meter provides a good opportunity to both reduce testing costs and increase operability in turbine engine ground test facilities.

1.1. Background

1.1.1. General Operation of a Bellmouth Flow Meter: The bellmouth flow meter is a type of head meter. A head meter brings about a change in static pressure of the flow. Based on the difference between the airflow total pressure upstream and a static pressure in the device, the flow rate can be determined by theoretical considerations.

The geometry and parameters for typical bellmouth flow meters are shown in Figure 2. The total pressure and temperature of the flow are typically measured at some

station in the upstream section of diameter D . The bellmouth itself has a contraction section which is typically composed of either elliptical or circular arcs. After the contraction follows a straight duct section which in a turbine engine test facility would lead to the engine face. The area of the straight section is referred to as the throat area and the throat diameter is denoted as d . The point of tangency of the contraction section and the straight duct is defined as Station 0. At some point downstream of Station 0, a measurement plane is defined at which the static pressure is determined. The mass flow rate for a bellmouth flow meter is determined based on the pressure measurements coupled with a bellmouth performance parameter.

1.1.2. Calculation of Discharge Coefficient: For compressible fluid flow an ideal flow rate, $\dot{m}_{ideal}$, can be defined as follows:

$$\dot{m}_{ideal} = p_T A M_{ideal} \sqrt{\frac{\gamma}{RT_T} \left(1 + \frac{\gamma-1}{2} M_{ideal}^2 \right)^{\frac{\gamma+1}{1-\gamma}}} \quad \text{Eq. 1}$$

Here p_T and T_T represent the total pressure and total temperature respectively. The ratio of specific heats is shown as γ and the ideal gas constant is represented as R . The flow area, A , is the area of a plane normal to the axis of flow. Mach number, M_{ideal} , is defined by the ratio of the static pressure, p , and total pressure using the isentropic relation:

$$M_{ideal} = \sqrt{\frac{2}{\gamma-1} \left[\left(\frac{p}{p_T} \right)^{\frac{1-\gamma}{\gamma}} - 1 \right]} \quad \text{Eq. 2}$$

Equation 1 defines the ideal mass flow through the flow meter. The definition of the ideal flow through a bellmouth is based on the measured values of the upstream total

pressure and temperature and the throat static pressure. The area used in calculating the ideal flow corresponds to the cross-sectional area of the meter at the static pressure measurement plane. Therefore in this context the ideal flow represents the flow rate that would be indicated by a given static pressure measurement at a given measurement plane if the flow was a 1-D ideal flow.

In the use of an actual meter, the flow rate will differ from the ideal flow. Features of the flow such as boundary layers and flow streamline curvature will cause the actual flow rate to vary from the defined ideal flow. To obtain the actual flow rate in a given meter, the meter performance characteristics must be determined. This is accomplished through the use of the discharge coefficient or flow coefficient. The discharge coefficient is defined by the equation

$$C_D = \frac{\text{Actual mass flow rate}}{\text{Ideal mass flow rate}} \quad \text{Eq. 3}$$

Based on this definition, if the discharge coefficient is known it is possible to calculate the actual mass flow for both compressible and incompressible flow conditions. The use of the simple theoretical mass flow equations with a properly determined discharge coefficient allows for rapid easy computation of the actual flow rate. The expression of the theoretical flow rate for incompressible flow assumes constant density and is of the form:

$$\dot{m}_{\text{actual}} = A \left(\frac{C_D}{\sqrt{1 - \beta^4}} \right) \sqrt{2\rho_1(p_1 - p_2)} \quad \text{Eq. 4}$$

where β is the ratio of throat diameter to upstream diameter, d / D and subscripts 1 and 2 stand for the upstream and downstream measurement locations, respectively. The actual mass flow rate for compressible flow in a bellmouth meter will be

$$\dot{m}_{actual} = C_D p_T A M \sqrt{\frac{\gamma}{RT_T} \left(1 + \frac{\gamma-1}{2} M^2\right)^{\frac{\gamma+1}{1-\gamma}}} \quad \text{Eq. 5}$$

Calculation of the discharge coefficient requires a method of determining the actual flow rate. The methods to accomplish this can be broken into four primary categories.

The first method for determining the discharge coefficient is by passing a known mass of fluid through the meter. The fluid passes through the meter to a receiving volume. By measurement of pressure and temperature in the receiving volume, before and after mass is passed through the meter, the actual volume of fluid passing through the meter can be determined. This method works well for small meters but is limited by the size of available facilities. For larger meters with corresponding large flow rates, this method is impractical.

The second method is calculation of the mass flow through direct experimental measurement. Typically this method involves careful measurement of the flow field velocity and density at many stations across a cross-section of the meter. The mass flux distribution is then carefully integrated across the flow area to determine the total mass flow. This method requires laboratory time and equipment.

A third method is by comparison of the test venturi or bellmouth flow to the flow through a previously calibrated device such as a critical flow venturi. The bellmouth is mounted in series with a calibrated device and a steady flow rate is passed through both

meters. This has the disadvantage of adding the uncertainty of the calibrated device to the total mass flow uncertainty. In addition, for large meters it is also difficult to calibrate the reference venturi.

The final method of determining the actual mass flow rate is direct computation of the flow field. Using theoretical methods the flow field is calculated and numerically integrated to yield the actual mass flow. This method allows many configurations to be calculated rapidly, but should be verified if possible by one of the two previous physical methods.

The discharge coefficient is not one fixed value for a given meter. The discharge coefficient is defined properly in terms of a consistent process for a given flow meter. All the parameters of Equation 4 must be measured and obtained in a carefully defined manner. The definition includes the physical geometry of the meter, measurement locations, and measurement technique. Also the discharge coefficient is a function of flow variables such as Mach number and Reynolds number, inlet contraction ratio, inlet shape, and flow field nonuniformity in the entrance of the bellmouth.

It should be noted that the discharge coefficient does not represent an efficiency. Although the discharge coefficient is usually less than one, it is possible for it to be greater than one for a given configuration and measurement process. Furthermore, the discharge coefficient does not serve to make any sort of comparison between two different meters. Two different meters with different coefficients can give equally accurate measurements of the flow rate provided that their respective discharge

coefficients have been accurately determined and the required flow parameters are accurately measured.

1.2. AEDC Bellmouth Airflow Measurement

There are several variations in the design features of bellmouth flow meters installed in AEDC turbine engine test bellmouth installations compared to standard installations of flow measuring devices. At AEDC the inlet into the bellmouth flow meter contains a conical extension from the bellmouth inlet to the airflow supply ducting. This arrangement has been chosen to prevent the formation of undesired inlet vortices which adversely impact engine performance. In addition, there is no outlet to a larger duct. The bellmouth outlet is usually coupled directly to a duct of identical diameter leading to the engine face. In some cases, there may be an expansion or contraction section downstream of the meter to match the engine inlet diameter. Therefore, to obtain accurate flow measurements using the bellmouth meter requires a capability to determine the discharge coefficient for a given meter and a given meter installation.

Although it would be desirable for consistency to specify one defined bellmouth for use in engine test facilities, in actual practice a variety of bellmouths are used. Inlet bellmouths with varying profiles, lengths and contraction ratios can be encountered in typical operation. Measurement stations may be placed in different locations. Furthermore, facility installations and limitations may affect flow measurement locations.

It is therefore desirable to determine the value of the flow coefficient at a number of measurement locations other than the standardized locations.

The flow behavior of the bellmouth flow meter deviates from that of the critical flow venturi operation in that the bellmouth flow is sub-critical, typically in the range of 0.2 - 0.7 Mach number for turbine engine testing. The critical flow venturi, as a result of choked ($M=1.0$) operation, is isolated from flow disturbances downstream of the meter. This is not the case for the bellmouth where the subsonic flow allows the discharge coefficient to be affected by disturbances downstream of the measurement plane. This is particularly the case for installations in which the turbine engine face is located very near or even in the bellmouth.

Given these considerations there is a need for a capability to determine the flow coefficient for various bellmouth geometries and configurations. Since the flow rates required for turbine engine testing are often quite large (up to 1500 lbm/s), direct calibration of bellmouth meters is very difficult. Determination of the flow profile for each bellmouth under laboratory conditions would be costly, thus negating the stated purpose for using bellmouth flow meters instead of critical-flow venturis. Therefore there is a strong need to establish a computational capability for investigating and determining bellmouth flow meter discharge coefficients.

1.3. Statement of Problem

The problem addressed in this thesis consists of two parts. The first part concerns the computational determination of the discharge coefficient of a bellmouth flow meter isolated from any other facility effects. Such effects would include distorted inlet flow, surface defects, presence of struts and rakes, duct misalignment, or effects due to the presence of an engine. A method was developed by which the discharge coefficient of an arbitrary bellmouth flow meter can be computed for the flow conditions of interest.

Since the operation of altitude test facilities usually involves a direct connect mode of operation the range of conditions required for operation of the bellmouth meter is determined by the range of flow conditions at the inlet to the engine. For conventional turbine engine operation the approach velocity at the engine face is subsonic. Typical ranges for approach Mach number are 0.2 to 0.7. However, this represents the flow condition after diffusion or recovery process in the engine inlet or aircraft inlet duct. The flight condition may vary from low subsonic to supersonic conditions, dependent upon the flight envelope of the engine in question. The potential flight envelope also encompasses a large altitude range. The result therefore is that the bellmouth flow meter must operate over a large range of Reynolds number, from as low as 500,000 to as high as 40,000,000.

Based on these considerations we can state, in general, the requirements to compute the flow coefficient. The flow can range from a Mach of 0.2 to 0.7. Reynolds number may range from 0.5 to 40 million. The flow field is an internal compressible

viscous flow with boundary layer effects critical to proper determination of the flow coefficient. For typical installation and operation the flow field is axisymmetric and steady-state. Since multiple bellmouth installations are possible, it is required that the discharge coefficient be computed for multiple bellmouth geometries.

The second portion of this thesis dealt with the discharge coefficient of a bellmouth flow meter with a rotating blade row installed at the exit of the flow meter to simulate the effects of a downstream turbine engine on flow meter discharge coefficient. A computational method for assessing the impact of the engine proximity was developed. Since the blade row is rotating in the bellmouth discharge, a capability is required to calculate the upstream effect of the rotating fan blades on the bellmouth flow coefficient. The development of this capability was demonstrated and qualitative results of the impact of a sample engine face on the bellmouth performance are presented.

2. Literature Survey

2.1. ASME Long-Radius Flow Nozzle

2.1.1. Nozzle Description: The ASME long-radius flow nozzle is a standard device for flow measurement. The construction and use of this flow meter are detailed in [2],[3]. The flow meter, shown in Figure 2, consists of a flared tube which can be clamped between pipe flanges. The profile of the nozzle is essentially similar to the bellmouth inlet used in turbine testing.

The flared section provides a smooth entrance to the straight tube section which is defined as the throat. The intent of the contraction is to accelerate the fluid smoothly to the throat where it issues downstream into the pipe as a fluid jet of the same diameter as the throat.

The ASME long-radius flow nozzle is defined as one of two series: A low β series or a high β series, where β indicates the ratio of the throat diameter to the upstream pipe diameter, d/D . As shown in Figure 2, the low β series ($\beta < 0.5$) is defined by an elliptical inlet profile of fixed eccentricity in the contraction section. The profile of the high β series ($\beta > 0.45$) has a variable eccentricity which is a function of the upstream and throat diameter.

Flow measurement using the ASME nozzle relies on the use of an inlet and outlet pressure measurement. There are three basic configurations for measurement of the pressure differential, as shown in Figure 3.

The first configuration uses pipe wall taps. The inlet pressure tap is located on the pipe wall one pipe diameter upstream of the meter. The outlet pressure tap is located one and a half nozzle throat diameters from the inlet face of the nozzle for the low β series or one half diameter for the high β series. Flow exits the nozzle in a jet which reattaches downstream. The pressure tap locations specified are upstream of this reattachment and the pressure gradients at these locations are very small.

The second configuration makes use of a throat tap. The inlet tap is at the same upstream wall location as the wall tap configuration. The location of the outlet tap is in the throat section of the nozzle itself. The throat tap is usually located about 0.15 throat diameters upstream of the nozzle outlet. In this configuration, the throat tap must be connected through the wall of the outer pipe.

The third configuration is the corner tap configuration. In this configuration, pressure taps are placed in the corner of the nozzle flange and the pipe wall, on both the upstream and downstream side.

2.1.2. Discharge Coefficient: When the pressure difference is properly measured, then the flow rate through the nozzle can be determined through dynamic principles. The details of obtaining the flow rate of a ASME long-radius flow nozzle are described in Ref. [2].

Values for the discharge coefficient, C_D , are shown in Figure 4 and Figure 5 for ASME long-radius flow nozzles as presented in Ref [2]. Note that these values are based on the pipe diameter Reynolds number, Re_D , where $Re_D = \rho U d / \mu$. Values are given for a span of contraction ratio, β , of 0.2 – 0.8. Reynolds number ranges from 500 to 10^7 .

Figure 4 presents the discharge coefficient as a function of Reynolds number for various contraction ratios. The discharge coefficient ranges from a value of about 0.88 to a maximum of just over 0.99. The curve begins to flatten at higher Reynolds number. The increasing discharge coefficient corresponds with the relatively thinner boundary layer at higher Reynolds number. For an ideal flow, the boundary layer thickness is zero, corresponding to an infinite Reynolds number.

Figure 5 shows the discharge coefficient as a function of β for various Reynolds numbers. In general, the discharge coefficient decreases for larger contraction ratios. This is more pronounced at lower Reynolds number. However in a later release of the ASME standards [3], the various curves for β are collapsed somewhat by plotting versus the throat Reynolds number instead of the pipe Reynolds number. In this case Ref. [3] recommends several empirical equations for discharge coefficient based on pipe diameter, β , and throat Reynolds number.

2.2. Review of Incompressible Flow Meter Literature

Under ideal circumstances, each bellmouth in operation would be calibrated against a known standard prior to operation. However, in practical use, this is not achievable. For many smaller nozzles in non-critical operations, detailed calibration may not be economically justifiable. At the other end of the size range, large flow nozzles may not be calibrated because of the lack of adequate calibration facilities and the high cost of

such a calibration. Given these constraints, several authors have presented correlations for discharge coefficients based on theoretical considerations.

Shapiro and Smith [4] presented an early estimate for discharge coefficients while examining the friction coefficients in the inlets of smooth round tubes. Based on their studies of the apparent friction coefficient they proposed a discharge coefficient of the form:

$$C_d = \frac{1}{\sqrt{1 + 4\bar{f}_{APP}\left(\frac{x}{D}\right)}} \quad \text{Eq. 6}$$

Shapiro and Smith recommend a method by Langhaar [5] for calculating the apparent friction coefficient for a laminar inlet boundary layer and recommend an approximation for an effective x/D for the case of a bellmouth inlet.

Rivas and Shapiro [6] present one of the earliest theoretical models for discharge coefficient intended for flow meters with a smooth rounded entrance. Building on the expression given above by Shapiro and Smith [5], Rivas and Shapiro develop an expression for an equivalent length of pipe to represent the losses along the contraction section. Then traditional boundary layer theory is applied along an equivalent straight pipe length and the core flow is calculated as a potential flow solution. The results from this method gave good agreement to the ASME nozzle data up to a Reynolds number of about 1 million.

The authors point out that the development of this method allowed for important insights to be considered in flow meter design, construction, and installation. The availability of a theoretical model allowed for prediction of whether a particular contour

would have laminar, turbulent, or separated flow, and also provide insight as to the preferred location of the downstream pressure tap and the importance of surface roughness and initial turbulence.

Cotton and Westcott [7] presented a review of the manufacture, calibration, and performance of 17 nozzles built for conducting performance tests on large steam turbine-generator units. In discussing the manufacturing process, they emphasized the importance of maintaining careful control of the elliptical profile and especially the careful machining of the throat pressure taps. The importance of a clean square edge in the pressure taps was emphasized. A number of discharge coefficient curves from calibrations at three separate calibration facilities is presented along with a theoretical curve based on the equation presented above by Shapiro and Smith [4]. Very good agreement in results was seen between all three facilities. It was noted that in the regions where the nozzle boundary layer flow is turbulent, the calibration results were about .25 percent higher than the theoretical curve.

The existence of the turbulent boundary layer regime is shown [7] to be at throat Reynolds number greater than 1.3×10^6 , with a transition region between laminar and turbulent flow of 8×10^5 to 1.3×10^6 . Since the application of the nozzles of interest were in large scale steam-turbine performance testing, the higher Reynolds numbers were the range of interest. In this range, Reynolds number greater than 1.3×10^6 , the nozzles were determined to have a flow coefficient in the range of 0.994 to 0.998 with a slight increase with increasing Reynolds number. The authors noted that operation in the laminar and

transition region made it more difficult to obtain accurate, repeatable measurements resulting in considerable error.

In the peer review portion of the paper [7] considerable discussion is devoted to an improved theoretical model. As a result of the discussion the authors developed an extension of the method developed by Rivas and Shapiro [6] into the turbulent region. Also several peers commented on the need to account for static pressure tap size. The effect of static pressure tap size was accounted for according to the results published by R. E. Rayle, [8]. Application of this correction to the new theoretical curve resulted in a much improved comparison to the calibration data. The authors also comment based on theoretical considerations and experience that the effect of the distance from the throat taps to the nozzle exit face was probably not significant.

Hall [9] developed an analytical method for the determination of the discharge coefficient of cylindrical-tube orifices. Although not intended initially for flow nozzle application, it is significant because it introduces the formulation of the discharge coefficient in terms of the displacement thickness, which in turn is described as a function of Reynolds number based on length. This gives a discharge coefficient as a function of the geometry of the cylindrical-tube orifice and the Reynolds number for an effective diameter ratio of zero, where the throat diameter is very small relative to the upstream diameter.

In response to a shortcoming in the ASME standards of descriptions of flow nozzle operation in a plenum inlet, Leutheusser [10] presented a study on flow nozzles with zero beta ratio. Under the assumption that the flow in the upstream plenum is

irrotational, Leutheusser derived an analytical relationship for the displacement thickness. Based on the displacement thickness, he derived an expression for the discharge coefficient. The displacement thickness was evaluated as a function of Reynolds number using an approximation of the von Karman-Pohlhausen method for the computation of axisymmetrical incompressible boundary layers in accelerated flow. The resulting equation is a simple function of Reynolds number to be applied in the Reynolds number range of 1×10^3 to 1×10^6 and is intended only for laminar flow. Leutheusser obtained experimental data which showed good agreement over the Reynolds number range of 4×10^4 to 3×10^5 .

Soundranayaga [11] investigated the performance of two ISA metering nozzles. The performance of nozzles of zero area ratio and a nozzle having area ratio of 0.4 were compared. A zero area ratio nozzle is one in which the nozzle throat area is very small in comparison with the upstream area such as an inlet from a very large tank or an atmospheric inlet. The pressure distribution on the zero area ratio nozzles was measured over a Reynolds number range of 100,000 to 400,000. Experimental results indicated a laminar boundary layer formed at the bellmouth entrance, which separated to form a small separation bubble in the nozzle.

The velocity distribution within the boundary layer was measured by traversing the boundary layer with a total-head pressure probe. These measurements were used to determine the displacement area due to the boundary layer and in turn, the discharge coefficient. These readings were then used to determine the discharge coefficient at various nozzle Reynolds numbers.

The discharge coefficient was also calculated using theoretical considerations. The boundary layer was assumed to begin 1.4 throat diameters upstream of the nozzle exit, upstream of the rounded contraction of the nozzle. The laminar boundary layer was calculated using Thwaites method and then at the indicated point of separation, a turbulent boundary layer thickness was computed using the method of Truckenbrodt. Using this method, the variation of discharge coefficient with Reynolds number was computed. Reasonable agreement was obtained for Reynolds number of about 2×10^6 and greater. Soundranayagam concluded that the poor agreement at lower Reynolds number was due to transition being delayed past the calculated separation point and therefore the flow field was very different than the assumed model at those conditions.

Soundranayagam also investigated the performance of a 1.581 inch diameter ISA nozzle installed in a 2.5 inch pipe. This installation had an area ratio of 0.4. There were twelve pipe wall pressure taps upstream and twelve downstream of the nozzle.

It was discovered that the experimental discharge coefficient was sensitive to the quality of flow entering the nozzle. The initial runs were made with 40 diameters of pipe upstream of the nozzle. This showed a significant variation in discharge coefficient with varying Reynolds number. The runs were then made with 70 diameters of straight pipe upstream which resulted in much more consistent discharge coefficient.

The author also noted that there was a significant pressure gradient in the region of the throat pressure taps and therefore recommended that the pressure measurement be made at the throat of the nozzle itself for more consistent results. Finally the results of both nozzles were compared qualitatively. It was noted that the zero area ratio nozzle had

an inlet flow that approached as a “potential flow” whereas the 0.4 area ratio nozzle had a rotational, fully developed pipe flow velocity profile. This resulted in a different minimum pressure and therefore different adverse pressure gradients in the nozzle.

Benedict [12] presented a review of several suggested curve fits of discharge coefficient for ASME nozzles, both theoretical and empirical. The author proceeded to develop a new empirical relation based on the empirical data represented by the ASME flow nozzle discharge coefficient charts. The discharge coefficient is presented as a function of only the throat Reynolds number.

Cotton, Carcich, and Schofield [13] presented further findings based on experience with throat-tap nozzles. Results are presented based on data from 45 ASME throat-tap nozzles. Cotton, et al, presented a theoretical relation for discharge coefficient based on turbulent boundary-layer theory and using regression analysis on the calibration data to calculate a discharge coefficient equation. Since the correlation was based on turbulent theory, the authors suggest the expression be limited to Reynolds number greater than 3.675×10^6 . Based on the theoretical results it was noted that beta ratio has a small effect on the discharge coefficient, only a 0.19 percent variation for beta ratio range of 0.371 to 0.075.

The authors also took careful note of the need to correct the theoretical curve to account for the influence of pressure measurement errors due to the influence of pressure taps of finite diameter. Experimental results were presented which demonstrate a significant effect of the machining and manufacture of the pressure taps on the final discharge coefficient.

Miller and Kneisel [14] presented a comparison between flow nozzle laboratory data and the published ASME coefficients and several theoretical curves for discharge coefficient across a Reynolds number range of $1 \times 10^5 - 1 \times 10^7$. Their results indicated that, independent of Reynolds Number, for upstream pipe diameters less than four inches, the discharge coefficients were outside the published tolerance. The explanation of this discrepancy was unknown although it was speculated that relative surface roughness of the flow meter may have been a factor. Certainly for smooth surfaces, the flow coefficient should be independent of diameter for constant Reynolds numbers. Coefficients for beta ratios greater than 0.7 also fell outside the published ASME tolerance. However for pipe diameters greater than four inches and beta ratio less than 0.7 the coefficient could be predicted over a wider Reynolds number range than the current ASME published numbers.

Returning to theoretical considerations, Benedict and Wyler [15] presented a generalized expression for discharge coefficients for differential pressure type fluid meters. This generalized expression accounted for beta effect, nonuniform inflow velocity profiles, contraction effects and pressure losses between the taps. This general expression was then applied to each of several differential pressure type meters including the flow nozzle. Assumptions for the flow nozzle included an assumption of no contraction at the measurement station, and fully developed pipe flow at the inlet. Losses are accounted for by a loss coefficient including an assumed effective L/D and friction factor calculated by either assuming laminar or turbulent flow. Comparisons to earlier

theories and ASME calibration data indicate the new formulation is superior to some other theories but does not match the calibration data in all Reynolds number ranges.

Soon after this attempt, Benedict and Wyler [16] presented a comparison of discharge coefficient between the throat and pipe wall tap nozzles. The subject of the paper was which installation might be more accurate. Based on experiments involving both nozzle types the authors asserted that the pipe-wall taps were as accurate as throat-taps and sometimes more accurate in determining flow rate. In light of the easier construction and operation of wall-taps the authors recommend that pipe-wall taps should be considered where in many situations they might not be considered. In the discussion portion of the paper, others disagreed, pointing to the extensive experience and data base of throat-tap operation in comparison with the limited data-set presented by the authors.

Benedict and Wyler [17] again developed a theoretical discharge coefficient for incompressible flow. They combined a boundary layer solution with a potential flow solution for the core flow. For the boundary layer solution they used a stepwise, axisymmetric, laminar-turbulent boundary layer solution based on the method of Walz. Based on these formulations the authors derive an expression for discharge coefficient as a function of the displacement thickness, energy thickness, and bellmouth diameter.

The authors noted several important observations about the boundary layer. They noted that it is possible for several forms of boundary layer transition to occur. The effect of transition on discharge coefficient is to shift the discharge coefficient curve. For transition generated by a mild separation the discharge coefficient curve actually shifts upward, because such a boundary layer grows at a smaller rate than the corresponding

laminar layer. However for a more severe separation in which a separation bubble is formed, the boundary layer is thickened and the discharge coefficient is reduced. Benedict and Wyler also found that the nozzle profile shape does affect the discharge coefficient

Benedict and Wyler discuss the results of experiments from a plenum inlet ASME nozzle installation. Data were obtained for Reynolds number in the range of 10^5 to 10^6 . Over this range, the data showed good agreement with discharge coefficients calculated assuming a laminar boundary layer.

In a subsequent paper [18] Benedict presents a more specific relation for the case of a generalized flow meter in either a pipe or a plenum installation which has no flow separation. Based on boundary layer considerations he develops an expression for discharge coefficient in incompressible fluid as a function of the displacement thickness at the entrance and exit, the radius at entrance and exit, and the beta ratio. For a plenum inlet case, the expression simplifies to that given by Shapiro and Rivas.[6]

In a later reference [19], Benedict discusses further the discharge coefficient of an ASME elliptical nozzle installed in a plenum inlet. The discharge coefficient is given in analytical form as a function of displacement thickness, beta ratio and throat and inlet radius. The boundary layer displacement thickness is calculated numerically. Benedict notes that for Reynolds number below 10^6 a laminar boundary layer exists at the nozzle throat and for Reynolds number exceeding 10^6 a turbulent boundary layer exists at the nozzle throat.

Benedict discusses the importance of static pressure tap size on the measured discharge coefficient. A number of correlations are discussed which relate the error of static pressure measurement to various bellmouth parameters such as Reynolds number, tap size, and wall shear stress. Using the predicted tap error, the author predicts the measured discharge coefficient using finite tap sizes based on the theoretical solution with an effective zero area tap size.

Experimental data using water was collected from an ASME nozzle in a plenum inlet at Reynolds number ranging from 4×10^5 to 5×10^6 . Benedict drew several conclusions from the results. Above a Reynolds number of 10^6 there is not significant effect of tap size on the discharge coefficient of an ASME nozzle with a plenum inlet. This agreed with theoretical predictions that at higher Reynolds number the static tap correction factor approaches a constant. There was no significant difference between the experimentally determined discharge coefficient and the predicted discharge coefficient for Reynolds number about 3×10^6 . There was an increased scatter and increased value for the discharge coefficient below Reynolds number of 3×10^6 which the author proposes indicates the presence of a transition region between the boundary layer of the lower Reynolds numbers and the turbulent boundary layer of Reynolds number greater than 3×10^6 .

The author proceeds to extrapolate these calculations to Reynolds number up to 40×10^6 . Benedict explains this is necessary due to the fact that calibration facilities are limited to test at throat Reynolds numbers up to a maximum of about 5×10^6 .

Murdock and Keyser [20][21] present a method for the extrapolation of calibration data of ASME Power Test Code (PTC) throat tap nozzles[3]. The ability to extrapolate accurately is a critical issue since for larger Reynolds number nozzles, calibration facilities are not available. The foundation of the method is the development of an expression for discharge coefficient as a function of Reynolds number. The derivation is based on basic boundary layer theory and assumes the flow to be in one of four possible regions, wholly laminar, transition with laminar displacement thickness greater than turbulent, transition with turbulent displacement thickness greater than laminar, and wholly turbulent. The result of this development matched the PTC data very closely. The authors note that the relationship derived indicates that the flow remains in transition up to a Reynolds number of 2×10^6 , much higher than previously suggested. The expression is recommended by the authors for Reynolds number greater than 1×10^6 .

2.3. Performance of Flow Meters in Compressible Flow

In comparison with the incompressible results presented above, little published literature is available on the performance of flow meters with compressible effects taken into account.

The ASME standards referenced earlier [2] [3] specify calculation of compressible fluids through the use of an expansion factor, Y_a . The expansion factor is derived from reversible adiabatic assumptions and accounts for variations in density based on the pressure ratio and diameter ratio of the nozzle. The expansion factor is

multiplied by the hydraulic equation for mass flow to yield a mass flow corrected for the compressibility. This method assumes that the discharge coefficient remains unchanged for a given Reynolds number on the basis of similarity.

Smith [22] presents data on the calibration of a small bellmouth inlet. The purpose of the inlet was to measure compressor airflow in propulsion simulators. The bellmouth of the study had a throat diameter of 3" and was operated over a Mach number range of 0.15 to 0.5 and Reynolds number less than 1 million. Results of the calibration indicated a discharge coefficient that varied from 0.966 to 0.972. The author comments that primary variation was with Mach number and that the effect of Reynolds number was negligible. This is contrary to what would be expected based on previous references. The range of Reynolds number would lead to an expectation of a transition regime in which the Reynolds number was a strong effect. The cited coefficients are also lower than the coefficients for an ASME flow nozzle.

Reimer [23] details a method of determining flow coefficient of ASME flow nozzles by thrust measurement. This method describes a theoretical relation between the discharge coefficient, the velocity coefficient, and measured thrust. The velocity coefficient is defined as the ratio of the outlet velocity and the ideal exit velocity based on the pressure ratio of the nozzle. The product of the velocity coefficient and the discharge coefficient should be slightly less than unity which is the case for ASME flow nozzles. This method was developed for use in large meters in aircraft-jet nozzles where primary methods of calibration are difficult because of the large size of the meters.

Reimer also comments on observations of nozzles used to prove nozzle thrust facilities. The following observations were noted. 1. The flow coefficient of an ASME nozzle flowing air at high pressure ratio is lower than the published ASME values for incompressible flow at the same Reynolds number. 2. The flow coefficient of an ASME nozzle flowing air at high pressure ratio increases with throat diameter independently of Reynolds number. It should be noted that this should not be the case if similarity holds. However the discharge coefficient is primarily a function of displacement thickness. Therefore unless displacement thickness grows exactly linearly, the relative displacement thickness for different scale nozzles will not be identical. It was also determined that the surface finish and polish of an ASME nozzle significantly change the flow coefficient. Note that these operating conditions were typically of higher pressure ratio and Mach number than those in an engine test facility inlet air stream.

Smith and Matz [24] compared the performance of an 8-inch ASME nozzle operating at compressible and incompressible conditions. The nozzle was tested and calibrated over a range of Mach numbers from 0.4 to critical flow and over a range of Reynolds numbers from 5×10^5 to 4×10^6 . The nozzle was also calibrated in incompressible flow using water over a Reynolds number range of 1.7×10^5 to 1.6×10^6 .

They noted a variation of discharge coefficient with subsonic Mach number at constant Reynolds number in compressible flow. This variation was within about ± 0.5 percent of the ASME recommended value. Also the difference between the incompressible flow and the lowest Mach number tested (0.4) was more than one percent. The incompressible performance of the nozzle was less than the ASME recommended

value but was within .7 percent of it. This study suggests that significant differences exist between the recommended ASME discharge coefficient values and the actual discharge coefficient of a nozzle operating at compressible flow conditions.

Smith and Matz noted that there was strong evidence of boundary layer transition effects in the compressible results. Also of significance was the manner in which the wall static pressure taps were constructed. They note that large wall static taps can result in an overpressure effect and therefore concluded that proper handling of static pressure measurement is critical to proper measurement and interpretation of nozzle coefficient data.

Smith & Matz also developed a theoretical model using the computer program developed by Wehofer and Moger [25]. Model results were obtained for both an inviscid and boundary layer modified solution. They noted that strong coupling occurs between the viscous effects in the nozzle and the nonviscous effects. The presence of the boundary layer in the model shifts the Mach number distribution in the throat region. They also point out the importance of the discontinuous wall curvature where the elliptical inlet section attaches to the cylindrical throat region. This leads to rapid diffusion accompanied by a thickening of the boundary layer and perhaps even flow separation.

Lahti and Hamed [26], [27] present an investigation into the verification of a theoretical discharge coefficient for an optimized subcritical airflow meter. Lahti develops a theoretical discharge coefficient using two computer codes, one to calculate the core flow outside the boundary layer and one to calculate the boundary layer itself.

The bellmouth meter used was not an ASME standard profile, but rather a specially designed bellmouth for use in aircraft engine ground testing.

Lahti notes in [26] the sensitivity of the flow measurement to errors in pressure measurement, especially at lower Mach numbers. Therefore he recommends an optimized profile for which the area at the measurement station be reduced to the minimum allowable without risking choked flow, thereby reducing the pressure measurement error.

Lahti and Hamed present the results of the theory in comparison with data taken from the recommended nozzle calibrated against a critical flow ASME nozzle. They achieved excellent agreement with wall static pressure distributions and the theoretical predictions of discharge coefficient were within 0.25% of the calibration standard.

3. Calculation of Bellmouth Discharge Coefficient

3.1. Description of Engine Facility Bellmouth

Accurate measurement of flow using any type of meter requires careful observance of the prescribed installation and measurement methodology. For example, the discharge coefficients specified by the ASME for flow nozzles are valid only for specified measurement locations and installation. In actual applications it may be desirable or necessary to locate pressure tap measurement at other locations.

Of particular interest to this study is the application of a bellmouth flow meter in the measurement of airflow in an engine test facility. The meter used in such a facility must of necessity vary from the standard flow nozzles specified in the literature such as the ASME or ISA nozzles. There are several differences between an engine facility bellmouth and the ASME long radius flow nozzle. On the upstream side the elliptical contraction section is connected by a conical extension all the way to the upstream duct diameter. This extension is required due to flow quality considerations. It has been determined [28] that such an extension will prevent the formation of undesirable inlet vortices.

On the downstream side, following the contraction section, there is no nozzle outlet to a duct of upstream diameter. The bellmouth contraction connects directly and smoothly to the downstream duct. This downstream duct connects directly to the engine

inlet. For engine diameter matching, a contracting or expanding section of duct may be required between the bellmouth and the engine face.

It is desirable to be able to determine the discharge coefficient for the bellmouth meter for a variety of measurement locations. Different testing configurations may require measurements at different planes and may also affect the optimal measurement location. Therefore it was undertaken to apply a computational fluid dynamics (CFD) technique to allow for determination of the discharge coefficient over a range of bellmouth configurations and flow conditions.

3.2. WIND CODE Description

The WIND code is a product of the NPARC Alliance, a partnership between the NASA Glenn Research Center and the Arnold Engineering Development Center. The Boeing Company has been closely associated with the NPARC Alliance and represents the interests of the NPARC User's Association.

The WIND code is a structured, multi-zone, compressible flow solver. It contains several turbulence and chemistry models. WIND is a merger of the capabilities of three existing CFD codes – NPARC, (the original NPARC Alliance flow solver), NXAIR (an AEDC code used primarily for store separation problems), and NASTD (the primary flow solver at McDonnell Douglas, now part of Boeing). Joint development began in early 1997. The current version is WIND Version 4.0.

The WIND code can be run in either serial mode on a single processor or in parallel mode on multiprocessor machines or multiple machines connected by a network. Parallel mode is accomplished by splitting the flow into multiple zones with each zone being assigned to specific processors.

The WIND code uses structured zonal grids. Zonal boundaries can be either abutting or overlapping. Grid generation is accomplished by external grid generation programs which are then stored in a Common Grid file (.cgd) format along with descriptions of the boundary condition types and zonal connectivity data.

The input data file is a text file created by the user which uses descriptive keywords to specify information about the flow problem and numerical solution options. There are two primary output files, the Common Flow file (.cfl) which contains the flowfield solution, and the List Output file, a text file which contains information about the computer run, including convergence information such as residuals and any error messages generated in the course of the run.

The WIND code contains a variety of physical models relevant to modeling internal fluid flow including the following capabilities:

- 2D, 3D, axisymmetric, or quasi-3D steady or unsteady flow.
- Euler, Navier-Stokes, thin-layer Navier-Stokes, or parabolized Navier-Stokes equations.
- Algebraic, one-equation, and two-equation turbulence models.
- Perfect gas, frozen chemistry, equilibrium air, and finite rate chemistry models.

The WIND code also contains a variety of numerical methods which include the following capabilities:

- Explicit, scalar implicit, or block implicit solution operator with time-accuracy via a Runge-Kutta or Global Newton algorithm.
- Central, Coakley upwind, Roe upwind, and physical upwind explicit differencing, 1st to 5th order.
- Implicit and explicit boundary conditions
- 2nd- and 4th-order explicit smoothing, boundary damping, and TVD flux limiting.
- Time step specification through a global constant time step or by a local CFL number
- Convergence acceleration using grid sequencing, local CFL numbers, and ramped CFL numbers.

3.3. Description of Bellmouth Test and Data Set

Recently a number of experiments were performed at AEDC research test cells by Beale, et.al. [29], to provide data for the purpose of exploring the bellmouth discharge coefficient and to provide data in support of validating CFD results. The general experimental configuration is shown in Figure 6. The research test cell provides for mounting two flow measurement devices in series. Upstream of each flow measurement station is an instrumented plenum with flow straightening devices. The facility was subjected to vacuum and pressure leak tests to assure that the leakage between the two

bulkheads was negligible. In the upstream bulkhead, a calibrated critical flow venturi was installed to serve as an airflow reference for the test bellmouth installed at the downstream bulkhead.

The facility was equipped with various kinds of instrumentation for measuring the flow entering and in the throat of the bellmouth meter. Instrumentation included boundary layer total pressure rakes and a transverse static pressure probe to measure the bellmouth throat core flow. Total temperature and pressure were measured upstream of the bellmouth in the plenum section. This instrumentation allows for determination of the bellmouth flow by integrating the flow field across the bellmouth under the assumptions that static pressure at the wall remains constant through the boundary layer and the total temperature remains constant at the plenum value. Therefore the total system provides two primary methods of determining the bellmouth flow. One, by comparison with a calibrated venturi and two, by direct integration of the flow field.

The experimental bellmouth geometry is illustrated in Figure 7. The bellmouth profile is that specified for the ASME low-beta (high contraction) bellmouth design. The specifications [2],[3] for low-beta design require an elliptical contraction section with the semi-major axis equal in length to the throat diameter and the semi-minor axis equal in length to two-thirds of the throat diameter. The low-beta design also specifies a straight section equal to 0.6 times the throat diameter. For a throat pressure tap the recommended location is 0.2 diameters upstream of the exit plane as shown in Figure 2. Figure 8 shows the installed configuration of the bellmouth. The test bellmouth diameter was 16 inches and the upstream plenum diameter was 35.25 inches. Note the conical extension from the

bellmouth to the plenum wall. As mentioned earlier, this is for the purpose of controlling undesired inlet vortices.

Figure 9 shows the boundary layer probe installation and detail. The boundary layer was measured at two axial stations: 3 inches downstream of the “throat” or tangent point (Station 3) and 8 inches downstream of the throat (Station 8). Each rake contained thirteen total pressure probes. Also a number of wall static pressure taps were installed longitudinally along the bellmouth inlet as shown in Figure 10. At 3 longitudinal station, taps were also installed around the circumference as shown. More detail on the experiment and instrumentation is available in Ref. [29].

3.3.1. Examination of Data: Experimental data from the boundary layer total pressure rakes are presented in Figures 11-18. The total pressure measurements obtained from the total pressure rakes have been normalized by the upstream total pressure. Each of the figures contains data for five Reynolds numbers at a specific Mach and bellmouth station.

Figures 11-14 show experimental data for Station 3 (see Fig. 8) at throat Mach numbers of 0.31, 0.4, 0.5, and 0.7 respectively. Reynolds number was increased by increasing total pressure. It is immediately apparent that the case of Reynolds number of 1×10^6 has a different character than the other Reynolds number conditions. This range of Reynolds number is squarely in the region described in the literature [2,3,6,7,17,19,21,22,24] as the transition zone from laminar to turbulent flow which ranges from 5×10^5 to 3×10^6 . The data at Station 3 indicate that the boundary layer at Reynolds number 1×10^6 is thinner than the other cases. The cases at higher Reynolds

number suggest a turbulent boundary layer with corresponding Reynolds number variations.

Figures 15-18 show boundary rake experimental data for Station 8 at the previously identified Mach numbers. Once again, the profile at a Reynolds number of 1×10^6 stands out from the remainder. However, it is similar in profile to the remaining curves. This suggests that at Station 8 that the profile has transitioned to turbulent flow but remains thinner because of the delayed transition. The remaining profiles appear in the order expected for turbulent flow. For increasing Reynolds number, the boundary layer should be thinner. In examining Figures 15-18 it is noted that in general, the case of Reynolds number = 2×10^6 appears to have the thickest profiles, with the boundary layer thinning for progressively larger Reynolds number.

The data at Station 8 also exhibit an anomaly at the 7th probe from the wall. Beale, et al [29] have not been able to provide a totally satisfactory explanation after numerous cross checks and instrument examinations. It is believed that the anomaly may be related to a small backstep at the junction between the conical extension and the bellmouth. The backstep was present in the experiment but not modeled. A full investigation of the anomaly using additional measurements and backstep computations has yet to be conducted. The behavior exhibited at Station 8 is present across all Mach and Reynolds numbers for which boundary layer data were gathered.

Some of the earlier theoretical attempts [9] [10] to characterize the discharge coefficient were based on the displacement thickness. To a first approximation the discharge coefficient can be expressed as $C_d = 1 - 4\delta^* / D$, assuming the flow outside the

boundary layer is uniform. Attempts were then made to characterize the displacement thickness as a function of Reynolds number and other flow parameters.

The displacement thickness was calculated based on the experimental boundary layer data just presented, assuming a constant static pressure and constant total pressure, using the compressible displacement thickness formulation. The measured displacement thickness is shown in Figure 19 for Station 3 and Figure 20 for Station 8. The curves are more uniform and have a wider range of values at Station 3 than at Station 8. At both stations, the displacement thickness is seen to be significantly smaller at Reynolds number of 1×10^6 than the other Reynolds numbers, indicating again the transition region.

Discharge coefficients based on displacement thickness are shown in Figures 21 and 22. The same general comments apply to these figures as the previous two. They are presented for the purpose of comparison with computed discharge coefficients to be discussed in a subsequent section.

3.4. WIND Computation of Bellmouth Flow

3.4.1. Description of Grid: The grid used for computation of the bellmouth flow is shown in Figure 23. This grid contains 201 points in the i index and 121 points in the j index. The grid is packed along the duct wall with a y^+ of about 1-9 along the wall for various flow conditions. The duct wall was set to a no-slip wall condition. The centerline of the duct was modeled as a symmetry boundary, assuming axisymmetric flow in a duct of circular cross section. On the inlet boundary, the conditions set were fixed total

pressure and temperature. The outlet condition was specified by setting a constant outlet boundary pressure.

Flow conditions were established by setting the proper inlet and outlet conditions. For all cases, the total temperature was set to 530 R to match the experimental condition. Mach number was set by specifying the proper isentropic ratio of total pressure at the inlet to static pressure at the nozzle outlet. The Reynolds number was set by specifying the proper total pressure in conjunction with the Mach number. A listing of the flow conditions for the various cases is given in Table 1, Appendix A.

Determining the discharge coefficient from the CFD solution requires both a numerical integration of the mass flux and a determination of the theoretical flow. The discharge coefficient was earlier defined as:

$$C_D = \frac{\text{Actual mass flow rate}}{\text{Ideal mass flow rate}} \quad \text{Eq. 3}$$

The actual flow rate is determined by numerically integrating the mass flux across a fixed plane in the duct. Therefore the expression for discharge coefficient assuming compressible flow becomes

$$C_D = \frac{\iint \rho v_x dA}{p_T A \sqrt{\frac{\gamma g_c}{RT_T} \frac{2}{\gamma-1} \left(\frac{p}{p_T} \right)^{\gamma+1/\gamma} \left(\left(\frac{p}{p_T} \right)^{1-\gamma/\gamma} - 1 \right)}} \quad \text{Eq. 7}$$

For an axisymmetric solution the actual flow is evaluated as:

$$\dot{m}_{actual} = \iint \rho v_x dA = 2\pi \int_0^{r_{wall}} \rho v_x r dr \quad \text{Eq. 8}$$

For a solution with density and velocity stored at n nodes mass flux was calculated as

$$\dot{m}_{actual} = \sum_{j=1}^{n-1} \left(\frac{(\rho v_x r)_j + (\rho v_x r)_{j+1}}{2} \right) (r_{j+1} - r_j) \quad \text{Eq. 9}$$

To check for adequate grid resolution several test calculations, called “runs,” were performed with a half-resolution grid (61x101) and a double resolution grid (241x401) in comparison with the nominal resolution of 121x201. Cases were run for Reynolds numbers of 3×10^6 , 5.75×10^6 and 20×10^6 for a Mach number of 0.5. For constant Reynolds number of 5.74×10^6 , cases with Mach numbers of 0.31, 0.5 and 0.7 were calculated.

Computed results for Mach =0.5 and Reynolds number of 5.75×10^6 are shown in Figures 24 – 27. The wall static pressure in the throat regions is displayed in Figure 24. The pressures computed with the nominal grid and the doubled resolution grid are seen to be very close, with the results computed with the half resolution grid somewhat lower. At the exit plane a pressure discontinuity is seen on the wall. This small error in pressure in the WIND code is seen only at the wall and outlet boundary. It does not extend across the duct exit and does not affect the outlet pressure.

Despite being very close in pressure there is some difference between the nominal and doubled resolution grid in the discharge coefficient solution obtained by integrating the entire flow field, shown in Figure 25. In this figure the half resolution results are closer but below the nominal resolution results. Since the wall pressure results were the opposite, this indicates that there are differences in the mass flux. The mass flux in the boundary layer is shown in Figure 26 and shows insignificantly higher mass flux in the boundary layer for the double resolution grid. Mass flux in the core flow is shown in Figure 27. In this figure, the mass flux of the double resolution grid is slightly smaller

than the nominal grid in the outer diameter, with the values crossing at about a 3 inch radius.

The variation of the computed discharge coefficient with respect to grid resolution exhibited in the throat region is approximately 0.07 percent. Since the desired accuracy for the bellmouth meter is 0.5% these results are seen to be very close. For the purpose of examining the discharge coefficients across various Mach and Reynolds numbers, the nominal grid is judged to be sufficient. If accuracy requirements were increased for a particular bellmouth flow meter, the use of a higher resolution grid may be warranted. Discharge coefficient comparisons at several Mach and Reynolds number combinations are shown in Figures 28-31.

3.4.2. Turbulence Model: The WIND code contains several turbulence models. These models fall into three broad categories; algebraic models, one-equation models, and two equation models. The selection of the proper turbulence model is critical because of the dependence of the flow coefficient on the boundary layer thickness. For well-designed nozzles the primary difference in flow rate between the ideal flow and the actual flow is in the momentum deficit of the boundary layer. Therefore it is important to select the turbulence model that best matches the experimental data.

Three turbulence models most commonly used in WIND were selected for comparison to the experimental data, one from each of the aforementioned categories. The algebraic model selected was the Baldwin-Lomax (Lomax) [30] model. The one-equation model selected was the Baldwin-Barth (Barth) [31] model. Finally, the two-equation model selected was the Menter Shear Stress Transport (SST) [32] model. The

SST model uses the $\kappa - \omega$ model near the wall and the $\kappa - \varepsilon$ away from the walls. Results for all three models were calculated for Mach number = 0.5 and Reynolds numbers of 2×10^6 , 3×10^6 , 3.9×10^6 and 5.77×10^6 .

The experimental data of Beale, et al [29] and the total pressure profiles calculated in the present work at Station 3 are compared in Figures 32-35 for different Reynolds number. The Barth model yields predictions that overshoot the inner layer and then cross the experimental data to undershoot the outer layer. The SST and the Lomax models predict profile shapes that are very similar. Both overshoot the inner layer slightly and then undershoot the outer layer slightly with the SST model showing a sharper “turn” towards the boundary layer edge. The comparisons improve as the Reynolds number increases.

Predictions of total pressure profiles for Station 8 results are shown in Figures 36-39 for different Reynolds numbers. All of the models appear to give reasonable predictions. The Barth model appears to match the outer layer well but overshoots the inner layer somewhat. The SST and Lomax models match the inner layer very well and are nearly coincident with the Barth model on the outer layer. Evaluation at this station is made more difficult by the behavior of the experimental data points in the middle of the boundary layer that seem to be somewhat out of place. As mentioned earlier, the cause of this experimental profile anomaly is not known although it is perhaps related to the surface discontinuity upstream at the attachment point of the conical extension

The effect of the turbulence models on the comparison between experimental and theoretically predicted displacement thickness and momentum thickness was also

obtained. Figure 40 shows the obtained comparisons at Station 3. The experimental data at $Re = 2 \times 10^6$ may be in error because it is inconsistent with both the experimental and theoretical trends. Except for the case at Reynolds number of 2×10^6 , both the SST and Lomax models match the data well. The Barth model gives a lower value of displacement thickness than the data and the other models. Similar predictions for momentum thickness at Station 3 are shown in Figure 41. All three turbulence models consistently predict momentum thickness that is approximately ten percent higher than the data.

Station 8 displacement and momentum thickness comparisons are displayed in Figures 42 and 43 respectively. The SST and Lomax models are nearly identical at the first three Reynolds numbers. At Reynolds number 5.75×10^6 the SST appears a better fit. The displacement thickness calculated using the Barth model is consistently ten percent lower than the data and the other models. For the momentum thickness, the predictions made with the SST and Lomax models are about eight percent higher than the data, with the Barth model showing a better fit for this parameter.

Although a more detailed discussion of discharge coefficient will be presented shortly, a quick comparison of the calculated discharge coefficients at Station 8 obtained with the three turbulence models is presented in Figure 44. The difference in the calculated coefficient between the SST model and the Lomax model is very small, on the order of 0.01 percent, whereas the Barth model predicts a discharge coefficient that is consistently about 0.13 percent greater.

A number of cases were also run to compare the performance of each turbulence model when implemented at different grid resolutions. Predicted total pressure profiles

obtained with a half resolution (61x101) grid are presented in Figures 45 and 46 for stations 3 and 8 respectively, at a Mach number of 0.5 and a Reynolds number of 3.9×10^6 . Corresponding profiles are shown for a double resolution grid (241x401) in Figures 47 and 48. Comparison of these profiles to those obtained with the nominal grid resolution results, shown in Figures 34 and 38 show the results to be very similar. There does not appear to be a large effect of grid resolution on the predicted results at the resolutions calculated, for any of the turbulence models examined.

Based on the examination of these results it appears that both the two-equation shear stress transport model (SST) and the Lomax model provide reasonably good predictions of the data. The discharge coefficient is mostly affected by the displacement thickness. At station 8, the SST appears a slightly better fit, especially at the higher Reynolds number. Furthermore, the WIND documentation [33] recommends the SST model as more accurate in the presence of adverse pressure gradients. Since adverse pressure gradients were expected in certain regions of the nozzle, the SST model was chosen for the remaining bellmouth calculations.

3.4.3. Typical Results: Calculation of the flow in the bellmouth meter defined in Figure 7 was performed with the WIND code using the computational grid shown in Figure 23. The solution was performed using Roe's second-order flux-difference splitting algorithm [34], modified for stretched grids, and using a full block implicit scheme. For the 2-D axisymmetric cases to be presented, CFL varied from 1 to 10, with a CFL value of 5 typically used for the lower Mach numbers (0.2-0.4) and a CFL of 10 used for the

higher Mach numbers (0.5-0.7). In this section, results are presented for the case of Mach number = 0.5 and Reynolds number = 5.75×10^6 with a CFL value of 10.

Convergence history is shown in Figures 49 and 50. Convergence is monitored in the WIND code by examination of the max residual and the L2-norm of residual. The max residual history is shown in Figure 49. The residual curve is characterized by a rapid decay in magnitude until about 5,000 iterations and then a slower drop to 15,000 iterations. The L2-norm is shown in Figure 50. It is very similar to the max-residual curve except it decreases to about an order of magnitude smaller than the max-residual with much less “noise” in convergence. Since the L2 norm represents an average of the residual of the entire solution, this value was examined to determine convergence.

The calculated mass flow vs. station is presented in Figure 51. The calculated mass flux is the value to be used as the actual flow in the discharge coefficient calculation. For a perfect steady state solution the calculated mass flow at each station would be a constant. Variations from a constant value are caused by error in the solution and also error in the interpolation and integration method used to calculate the mass flow. The location of the sharp peak in station mass flow shown in Figure 51 corresponds to the corner in the inlet plenum. At this location there is a wall discontinuity and the grid is inclined relative to the vertical line of integration, increasing the amount of interpolation necessary. Aside from this sharp peak the mass flow is seen to reside in a band of about 0.036% of the average value. Within the throat region from Station 0 to the exit, the grid lines are exactly vertical and the calculated mass flow is seen to be constant.

Figures 52-55 presents several contour plots for a typical solution. Results are for a throat Mach number of 0.5 and a throat Reynolds number of 5.75×10^6 . Noted on the plots are the locations of Station 0, Station 3 and Station 8. Station 0 is defined as the point of tangency between the elliptical contraction section and the straight throat section. In other words, Station 0 marks the beginning of the straight section and will be referred to as the throat. Station 8 corresponds to the ASME recommended location of throat tap placement at one half the throat diameter.

Figure 52 presents the static pressure contours. Of particular interest is the region near the wall at the throat. This area exhibits a localized region of lower pressure which then recovers as it approaches Station 3. This low pressure region is present in all the solutions as a result of flow turning as the fluid is contracted and then enters the throat.

Figure 53 presents the Mach number contours. The Mach number in the entry region is very low, $M=0.0921$ at the center line for the case presented. In the engine test facility, the plenum acts as a stilling chamber. For the purposes of this study it is assumed that the stilling chamber provides bellmouth entrance flow that is free of turbulence.

Figure 53 shows that the corner marking the beginning of the conical inlet contains a small region of separated flow with recirculation. This recirculation is present in all the solutions, however does not seem to appreciably affect the discharge coefficient. The velocity in the region of the recirculation is very low, and is immediately followed by a region of rapid acceleration into the throat. The Mach number contour also exhibits the small “bubble” of overexpansion near the throat. The boundary layer, which will be presented in detail shortly, is just evident in the coalescence of the contours along

the top of the duct. For comparison purposes, the very similar velocity magnitude contours are presented in Figure 54.

Figure 55 presents the mass flux, ρv_x , contours. This figure is of interest because of the importance of accurately calculating mass flow. Determination of the discharge coefficient is dependent upon the actual mass flow as determined by the calculation. The mass flow was determined by numerically integrating the mass flux across the duct in the throat region.

Determining the discharge coefficient requires measurement of the static pressure at the wall because it is used in the calculation of the ideal mass flow. Figure 56 shows the calculated wall static pressure distribution along the length of the duct. The overexpansion near the throat is clearly evident. The minimum is seen to occur just before Station 0 with pressure recovery continuing for several inches. At about 5.5 inches downstream of the throat the pressure reaches a maximum value in the throat and then begins to fall again due to boundary layer thickening accelerating the inviscid core flow.

Also shown in Figure 56 is the ideal 1-D static pressure based on total pressure, Mach number and area ratio. It is important to note that the actual pressure is both greater than the ideal in some locations and less than the ideal in others. In those locations where the actual is greater than the ideal, the discharge coefficient based upon an ideal mass flow at that pressure will in general be larger than one.

The wall static pressure is used to calculate the theoretical flow used in the discharge coefficient calculation. Since the static pressure varies along the wall the

discharge coefficient is also a function of the pressure measurement location. This is seen clearly by examining the following equations.

$$C_D = \frac{\dot{m}_{actual}}{\dot{m}_{ideal}} \quad \text{Eq. 10}$$

$$\dot{m}_{ideal} = p_T A M_{ideal} \sqrt{\frac{\gamma}{RT_T} \left(1 + \frac{\gamma-1}{2} M_{ideal}^2 \right)^{\frac{\gamma+1}{1-\gamma}}} \quad \text{Eq. 11}$$

$$M_{ideal} = \sqrt{\frac{2}{\gamma-1} \left[\left(\frac{p_{measured}}{p_T} \right)^{\frac{1-\gamma}{\gamma}} - 1 \right]} \quad \text{Eq. 12}$$

Figure 57 shows the value of the discharge coefficient as a function of the wall static pressure measurement location. Notice the large variation of discharge coefficient upstream of the contraction. This illustrates the sensitivity of the ideal mass flow to measured wall pressure at very low speeds. We are interested however, in the discharge coefficient in the region just downstream of the contraction and throat as illustrated in Figure 58. Comparison of both Figure 57 and Figure 58 with the wall pressure in Figure 56 reveals that the discharge coefficient and pressure curves do not have the same shape. For example the discharge coefficient curve does not contain a minimum value in the same location as the static pressure curve and also is fairly constant in the contraction region between stations $x = -10$ inches and $x = 0$ inches. The discharge coefficient curve can be viewed as indicating how close the actual pressure compares to an ideal pressure.

Another interesting relation to examine is the axial distribution of the rate of change of the discharge coefficient, as shown in Figure 59. This is of interest because of instrumentation concerns. Since the discharge coefficient is specified for a specific

measurement location, then it is important when installing pressure taps to locate them accurately. Figure 59 illustrates how sensitive the discharge coefficient is to an error in pressure tap location. Since the discharge coefficient is typically very close to one, the slope value is effectively the error in mass flow measurement. Therefore, from Figure 59, if the tap were to be placed at Station 0, a 1 inch error in placement would result in about a 2.4% error in mass flow measurement, whereas a tap located at Station 5 would have perhaps an 0.05% error for a 1-inch error in placement.

It is interesting to note that there is another very flat region in the contraction section at a station of about -7.5 inches. This would seem to indicate that the pressure taps could also be located in the bell. However, there are some practical constraints to basing a discharge coefficient on this location. First, since the velocity is lower in this region than the throat there is an increased sensitivity to pressure measurement error. Secondly, the ideal flow calculation requires knowledge of the flow area. It is more difficult to measure the exact cross-sectional area in the contraction section than in the fixed radius of the throat region.

Predicted total pressure profiles in the boundary layer are presented in Figure 60 along corresponding experimental data set for both Station 3 and Station 8. Station 8 shows a thicker boundary layer profile than Station 3 as expected.

The boundary layer mass flux profiles are detailed in Figure 61 and Figure 62. These two plots serve to illustrate the two regions of flow and their influence on the actual mass flow. Figure 61 shows the mass flux in the boundary layer. The displacement thickness accounts for a large part of the difference between the ideal flow (which is

constant all the way to the wall) and the actual flow. This is the principal upon which the early attempts to calculate discharge coefficient as a function of displacement thickness were based.

The mass flux in the boundary layer at Station 3 is seen to be greater than at Station 8. Since the flow is constant at all stations, the integral of mass flux divided by the area must be equal. The resolution to this is seen in Figure 62 where at Station 8 the mass flux is higher in the core flow. This is partly accounted for by the increasing displacement thickness but the core flow may also be affected by the streamline curvature in the throat. In any case, proper determination of the ideal flow requires accurate calculation of both the boundary layer and the core flow regions.

3.4.4. Comparison with Experiment: A number of solutions were calculated using the WIND code for comparison to the experimental results. Solutions were run for Mach numbers of 0.31, 0.4, 0.5 and 0.7 at corresponding Reynolds number of 2×10^6 , 3×10^6 , 3.9×10^6 , and 5.75×10^6 . Comparisons to data are presented in Figures 63-78. Each plot contains the calculated results and experimental data for a given Mach number and Reynolds number at both Station 3 and Station 8.

In general the solutions are seen to provide a good fit to the data. At Station 3 in Figures 63-78, the CFD solution exhibits a slightly thicker boundary layer profile than the data for all cases. For Station 8 in Figures 63-78, the results are fairly good at both the inner and outer layer. The middle of the boundary layer is more difficult to judge at Station 8 due to the three points which exhibit a lower value than expected in comparison with the remainder of the data. As mentioned earlier, this is judged to be some type of

measurement error. This opinion is bolstered by the fact that the same discrepancy exists across all Mach and Reynolds number conditions.

Careful examination of the data at Station 3 in Figures 63, 67, 71 and 75 reveals that for Reynolds number of 2×10^6 the boundary layer appears to be thinner in comparison to the calculated profiles than for the higher Reynolds numbers. This is consistent with the flow nozzle at Reynolds of 2×10^6 operating in a boundary layer transition region. Further downstream at Station 8 (Figures 63, 67, 71 and 75) the results are better, indicating a fully turbulent flow. Since the WIND computation is based on a high-Reynolds number turbulent transport model (the SST model), application of results should be limited to turbulent flow, which for this data set seems to begin just above 2×10^6 .

Based on these favorable comparisons, the WIND code was used with the SST model to perform a number of bellmouth flow meter calculations described in the following sections.

3.4.5. Discussion of Computational Results: It is desired to characterize the discharge coefficient across the range of expected Mach and Reynolds number variation. Having known coefficients, the mass flow can be determined by selecting the proper value of discharge coefficient for a given Mach number and Reynolds number condition. With this in mind, solutions for the 201x121 grid shown in Figure 23 were run for the 40 conditions listed in Table 1. Based on these results, the variation of discharge coefficient with Mach number and Reynolds number has been theoretically obtained and tabulated.

Variation of the discharge coefficient as a function of Reynolds number for constant Mach Number is shown in Figures 79 and 80 for Station 3 and Station 8 respectively. As expected, the discharge coefficient increases with increasing Reynolds number, reflecting a thinning boundary layer. Overall variation of the discharge coefficient at Station 8 for these cases is about 0.65%.

The discharge coefficient curves for constant Mach number in Figures 79 and 80 are seen to have less variation at Station 8 than Station 3. This is illustrated more clearly by plotting the discharge coefficient as a function of Mach number for constant Reynolds numbers, shown in Figures 81 and 82. At station 3, Mach number is seen to have an effect on the order of 0.35% to 0.375% on discharge coefficient variation. However at Station 8 the variation is only 0.037% to 0.06%. At both stations, an increase in Mach number leads to an increase in discharge coefficient.

The effects of Mach number on computed discharge coefficients at various axial stations along the bellmouth are seen in Figure 83 at a Reynolds number of 5.75×10^6 . In the contraction section the variation in discharge coefficient across the range Mach number is seen to be as much as 2.5%. At station 0 the variation is about 1.5% and the variation decreases along the duct.

The ASME standards referenced in [2],[3] do not indicate a correction for Mach number. When calculating with a compressible fluid, ASME specifies the use of the compressibility correction factor Y which takes into account density variation with compressibility.

$$Y = \left[r^{2/\gamma} \left(\frac{\gamma}{\gamma-1} \right) \left(\frac{1-r^{\frac{\gamma-1}{\gamma}}}{1-r} \right) \left(\frac{1-\beta^4}{1-\beta^4 r^{2/\gamma}} \right) \right]^{1/2} \quad \text{Eq. 13}$$

where r is the upstream to throat tap static pressure ratio and β is the diameter ratio of the measurement device.

The results presented in this work indicate an effect of Mach number separate from the 1-D compressibility correction. In order to make this effect more clear, Figure 84 presents the ρv_x profile at Station 8 for a Reynolds number of 5.75×10^6 . In order to allow direct comparison of the profiles, each profile has been normalized by the ideal 1-D mass flux corresponding to the ideal flow used to calculate the discharge coefficient. The higher Mach number profiles are seen to be flatter and closer to the ideal flow, while for the lower Mach number curves, increased curvature is in the central inviscid core region of the flow.

It is clear from Figure 84, that the mass flux profiles are almost identical in the boundary layer. This is consistent with the expectation that the boundary layer should be primarily a function of the Reynolds number with little influence of Mach number at these lower Reynolds numbers. The difference is seen to be in the primarily inviscid core flow. At higher Mach numbers the profile is flatter and closer to the ideal flow, resulting in a higher discharge coefficient. The differences in streamline curvature are seen clearly in Figure 85. Contour plots of the mass flux, ρv_x , are shown for Mach numbers of 0.2, 0.5, and 0.7. The profile is seen to be much flatter for increasing Mach number. Also the size of the region of overexpansion in the throat (near Station 0) is seen to decrease

markedly with increasing Mach number. Since Station 3 is located in the later portion of this region, this accounts for the larger sensitivity of discharge coefficient to Mach number at Station 3.

In summary, the variation of discharge coefficient can be largely attributed to two factors. Discharge coefficient increases with decreasing boundary layer thickness, which is primarily a function of Reynolds number. The other factor is the effect of core flow streamline curvature which is primarily a function of throat Mach number. Increasing Mach number in general leads to a flatter mass flux profile downstream of the throat, and increasing discharge coefficient. As distance from the throat increases, the effect of curvature in the throat area decreases, the mass flux profiles all begin to flatten and the effect of Mach number begins to lessen as indicated by the results at Station 8 in Figure 83.

3.4.6. Comparison to ASME Flow Nozzle: Although the computational results compared well to the available data, it was of interest to also compare the computational results to the standard ASME flow nozzle. Although the installation investigated varies from that of the ASME standard, it seems reasonable that since the inlet contraction section is an ASME profile, the resulting discharge coefficients should be similar. Figure 3 presented the discharge coefficients of an ASME nozzle with pipe wall taps for various beta ratios. However, the configuration calculated more closely resembled the ASME throat-tap nozzle. In Reference [3], the ASME presents a typical calibration curve of a low-beta flow nozzle with throat pressure taps which is reproduced in Figure 86. In

absence of an actual calibration, Reference [3] recommends reading values off of Figure 86.

The discharge coefficient for ASME throat tap nozzles is based on a pressure tap location of one-half of the throat diameter downstream of the throat. This corresponds directly to Station 8. It is clear upon inspection that the values in Figure 86 are higher than the calculated values. The ASME values represent results for incompressible flow, therefore the best comparison should be the $M=0.2$ curve in Figure 80. The ASME recommended values are approximately 0.8 percent higher than the calculated values shown in Figure 80.

One source of this discrepancy is error in the wall static pressure measurement. The computational discharge coefficient results are based on calculated static pressure at the wall. The ASME results present the discharge coefficient as a function of the measured static pressure. It is known that static pressure tap measurements are subject to a measurement error from the interaction of the fluid flow with the lip of the static pressure tap.

This effect has been examined by several investigators. Rayle [35] presented a curve relating pressure measurement error to the dynamic head of the flow and the hole diameter. Later correlations by Franklin and Wallace [36] present the pressure correction as a function of the surface shear stress and the hole diameter. Franklin and Wallace determined experimentally the error in static pressure as a function of the shear stress at the boundary, the tap diameter, and local viscous properties at the wall. This nondimensional curve is reproduced in Figure 87. Benedict and Wyler [17] present an

expression for pressure tap correction based on the available data and using the formulation proposed by Franklin and Wallace.

Results for $Mach = 0.2$ were corrected using this method assuming a pressure tap diameter of $3/16$ of an inch. This diameter corresponds to the average value of the range of tap size ($1/8$ to $1/4$ of an inch) recommended by the ASME for throat tap nozzles. Results for this correction are shown in Figure 88. The adjusted discharge coefficient results are shifted upwards by 0.4% or greater for the given tap diameter.

3.4.7. Effects of Bellmouth Inlet Contraction Eccentricity: Since different bellmouth flow meters may have different inlet contraction profiles it is of interest to examine some variations in the inlet profile for possible influence on the discharge coefficient. Six inlet profiles, in addition to the ASME profile, are shown in Figure 89 for the elliptical portion of the flow meter. A number of profiles were developed by varying the length ratio of the major and minor axis while holding the length of the minor axis constant. The discharge coefficient for each of these profiles was calculated for $M = 0.5$ and Reynolds $No = 3 \times 10^6$.

As seen in Figure 90, there is a large variation in the axial distribution of the discharge coefficient in the contraction section of the nozzle for various axis ratios. The differences are smaller in the throat region however, as indicated in Figure 91. The higher axis ratio cases reach a maximum value before lower axis ratio cases. This results in the order of the curves inverting at around $x = 4.5$.

An indication of the overall effect of axis ratio is shown in Figure 92. The percent variation of discharge coefficient between an axis ratio of 0.3 and 1.0 is shown for

stations downstream of the throat plane. At the throat itself the variation is about 2.5% and decreases rapidly. At the recommended measurement station, Station 8, the variation is about 0.12%. The effect of axis ratio on discharge coefficient at Station 3 and Station 8 is shown in Figure 93. At Station 3 the discharge coefficient decreases with increasing axis ratio, while the reverse is true for Station 8.

3.4.8. Effects of Beta Ratio: The effect of contraction ratio, the upstream pipe diameter to throat diameter ratio, was also examined briefly. Based on earlier ASME results (Figures 4 and 5) for incompressible flow, the effect of beta ratio was expected to be small. This was indeed the case as can be seen in Figure 94 and 95. As the figure approaches the throat the discharge coefficients merge almost exactly. Figure 95 shows a small spread in values for a beta Ratio of 0.3555 to 0.6275 at Mach 5.

Discharge coefficients at various beta ratios for three Mach number values of 0.2, 0.5 and 0.7 are shown in Figure 96. The curve has a concave shape with the beta ratio of the original configuration at $\beta = 0.4539$ standing out with somewhat of a peak. However the magnitude of this peak even at $M = 0.2$ is only about 0.02% of the flow coefficient and is within the error of the flow calculation.

Overall, the effect of throat to plenum diameter ratio is seen to be quite small with the largest variation occurring at $Mach = 0.2$. The variation of discharge coefficient at this Mach number is only 0.00064 over the range specified.

4. 3-D Effect of Rotating Fan Installed in Bellmouth Flow Meter

4.1. Engine Proximity to Bellmouth Flow Meter

Measurement of mass flow in a turbine engine facility is potentially influenced by the presence of a turbine engine in the duct downstream of the flow meter. Often the engine is significantly downstream (up to 10 diameters) and probably causes little influence on the flow measurement. However for large turbofan engines, the engine inlet face can be in close proximity to the flow meter. The engine face is characterized by high speed rotating fan blades with tip speeds in excess of Mach 1.

The possibility exists that flow meter performance is impacted by flow distortion caused by the presence of the engine. Tip shocks, uneven flow distribution, and swirling flow are all possible and may influence the discharge coefficient. The present chapter uses CFD predictions to assess the importance of engine installation on bellmouth measurements.

4.2. Approach

At first glance the engine inlet flow appears dynamic and complex and very difficult to model computationally. However, the flow solution can be simplified by shifting the CFD frame of reference to the rotating fan. For steady rotation rate, the problem can be recast in terms of a stationary fan with a rotating inlet flow and rotating

walls as shown in Figure 97. This approach assumes that the first rotor, the fan, has the dominant effect on the upstream flow and that any stator row or other stages behind the first rotor can be neglected. In the rotating frame for constant rotation rate, the flow appears as a steady-state problem. By recasting the problem in the rotating frame therefore, a steady-state approach can be simulated and an unsteady computation can be avoided.

4.2.1. Navier-Stokes Equations in a Rotating Frame of Reference: In performing rotating machinery calculations in a rotating frame, there are two primary choices of coordinate system, cylindrical (radial) coordinates and rectangular coordinates. Typically, a radial coordinate system is more intuitive and straightforward for rotating calculations, however there are other considerations in this particular case. The Navier-Stokes equations for a rotating frame in rectangular coordinates are very similar structurally to the inertial frame. Therefore, it seems likely that a rotating capability could be added to a Navier-Stokes solver with less modification than transforming to an entirely different coordinate system. A comparison of the rotating and inertial forms of the Navier-Stokes equations in rectangular coordinates will indicate just what type of corrections must be made to implement a rotating calculation.

A comparison of the Navier-Stokes equations in the inertial and rotating frame was given by Ucer, et al [37] and is provided below. In this section the terms will be described and compared in more detail for the sake of clarity and discussion to follow.

Navier-Stokes Equation - Inertial Frame

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \quad \text{Eq. 14}$$

$$\frac{\partial}{\partial t} \rho \mathbf{v} + \nabla \cdot (\rho \mathbf{v} \otimes \mathbf{v} + p \bar{\mathbf{I}} - \bar{\boldsymbol{\tau}}) = \rho \mathbf{f}_e \quad \text{Eq. 15}$$

$$\frac{\partial \rho E}{\partial t} + \nabla \cdot (\rho \mathbf{v} H - k \nabla T - \bar{\boldsymbol{\tau}} \cdot \mathbf{v}) = w_f + q_H \quad \text{Eq. 16}$$

$$H = h + \frac{\mathbf{v}^2}{2} \quad (\text{stagnation enthalpy}) \quad \text{Eq. 17}$$

$$E = e + \frac{\mathbf{v}^2}{2} \quad (\text{total energy}) \quad \text{Eq. 18}$$

Navier-Stokes Equation - Rotating Frame

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{w}) = 0 \quad \text{Eq. 19}$$

$$\frac{\partial}{\partial t} \rho \mathbf{w} + \nabla \cdot (\rho \mathbf{w} \otimes \mathbf{w} + p \bar{\mathbf{I}} - \bar{\boldsymbol{\tau}}) = \rho [\mathbf{f}_e - 2\boldsymbol{\omega} \times \mathbf{w} - \boldsymbol{\omega} \times (\boldsymbol{\omega} \times \mathbf{r})] \quad \text{Eq. 20}$$

$$\frac{\partial \rho E^*}{\partial t} + \nabla \cdot (\rho \mathbf{w} I - k \nabla T - \bar{\boldsymbol{\tau}} \cdot \mathbf{w}) = w_f + q_H \quad \text{Eq. 21}$$

$$I = h + \frac{\mathbf{w}^2}{2} - \frac{\mathbf{u}^2}{2} = H - \mathbf{u} \cdot \mathbf{v} \quad (\text{Rothalpy}) \quad \text{Eq. 22}$$

$$E^* = e + \frac{\mathbf{w}^2}{2} - \frac{\mathbf{u}^2}{2} = E - \mathbf{u} \cdot \mathbf{v} \quad (\text{Rotational Energy Term}) \quad \text{Eq. 23}$$

$$\mathbf{u} = \boldsymbol{\omega} \times \mathbf{r} \quad (\text{Velocity of Rotating Frame}) \quad \text{Eq. 24}$$

$$\mathbf{v} = \mathbf{u} + \mathbf{w} \quad \text{Eq. 25}$$

The rotating frame rotates at a constant rate of $\vec{\omega}$ thereby imparting a velocity everywhere in the frame. The frame velocity, u is expressed by:

$$\vec{u} = \vec{\omega} \times \vec{r} \quad \text{Eq. 26}$$

This allows for the definition of a translation between rotating and inertial coordinates:

$$\vec{v} = \vec{u} + \vec{w} \quad \text{Eq. 27}$$

$$\vec{w} = \vec{v} - \vec{u} \quad \text{Eq. 28}$$

where $\vec{v}$ is the velocity vector in the inertial frame and $\vec{w}$ is the velocity vector in the rotating frame.

The frame velocity also has an impact upon variables containing kinetic energy terms. The total energy and total enthalpy in the inertial frame are:

$$H = h + \frac{v^2}{2} \quad \text{Total Enthalpy} \quad \text{Eq. 29}$$

$$E = e + \frac{v^2}{2} \quad \text{Total Energy} \quad \text{Eq. 30}$$

In the rotating frame two similar terms are defined.

$$I = h + \frac{w^2}{2} - \frac{u^2}{2} \quad \text{Rothalpy} \quad \text{Eq. 31}$$

$$E^* = e + \frac{w^2}{2} - \frac{u^2}{2} \quad \text{Rotating Energy Term} \quad \text{Eq. 32}$$

Using the relation between rotating and inertial velocities these terms can be related as:

$$I = H - \vec{u} \cdot \vec{v} \quad \text{Eq. 33}$$

$$E^* = E - \vec{u} \cdot \vec{v} \quad \text{Eq. 34}$$

Using these relations, the following comparisons and comments are made.

The continuity equation is unchanged

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \quad (\text{Inertial}) \quad \text{Eq. 35}$$

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{w}) = 0 \quad (\text{Rotating}) \quad \text{Eq. 36}$$

The momentum equation contains additional source terms in the rotating frame formulation:

$$\frac{\partial}{\partial t} \rho \mathbf{v} + \nabla \cdot (\rho \mathbf{v} \otimes \mathbf{v} + p \bar{\mathbf{I}} - \bar{\boldsymbol{\tau}}) = \rho \mathbf{f}_e \quad (\text{Inertial}) \quad \text{Eq. 37}$$

$$\frac{\partial}{\partial t} \rho \mathbf{w} + \nabla \cdot (\rho \mathbf{w} \otimes \mathbf{w} + p \bar{\mathbf{I}} - \bar{\boldsymbol{\tau}}) = \rho [\mathbf{f}_e - 2\boldsymbol{\omega} \times \mathbf{w} - \boldsymbol{\omega} \times (\boldsymbol{\omega} \times \mathbf{r})] \quad (\text{Rotating}) \quad \text{Eq. 38}$$

In the inertial frame, the term $\mathbf{f}_e$ represents any body forces present. The two additional terms on the right hand side of the rotating frame expression are the Coriolis acceleration ($-2\boldsymbol{\omega} \times \mathbf{w}$) and the centrifugal acceleration ($-\boldsymbol{\omega} \times (\boldsymbol{\omega} \times \mathbf{r})$). These terms are additional body forces present in the rotating frame formulation.

The energy equation does not contain additional terms but note the replacement of the energy and enthalpy terms by their rotational counterparts.

$$\frac{\partial \rho E}{\partial t} + \nabla \cdot (\rho \mathbf{v} H - k \nabla T - \bar{\boldsymbol{\tau}} \cdot \mathbf{v}) = w_f + q_H \quad (\text{Inertial}) \quad \text{Eq. 39}$$

$$\frac{\partial \rho E^*}{\partial t} + \nabla \cdot (\rho \mathbf{w} I - k \nabla T - \bar{\boldsymbol{\tau}} \cdot \mathbf{w}) = w_f + q_H \quad (\text{Rotating}) \quad \text{Eq. 40}$$

These transforms, while straightforward, provide a critical roadmap to understanding proper solution techniques in the rotating frame.

It can be seen by examination that the equations in the rotating frame are in basically the same form as the inertial frame. The only exception is the addition of two source terms on the right hand side of the momentum equation. The attractiveness of using the Navier-Stokes equations cast in the rectangular form instead of cylindrical polar form is now apparent. For a Navier-Stokes computer code developed in the inertial frame for rectangular coordinates, it would seem that calculation in the rotating frame could be accomplished with the same structure and algorithms, with the addition of the two source terms and the modification of just a few parameters.

4.2.2. WIND Code Advertised Capabilities: As described in the users manual [33] the WIND code contains a rotating frame of reference capability. This is a newer capability which has not been exercised in depth. The rate of rotation and the rotational axis are defined as inputs to the solution. This capability should allow calculation of the flow field in and through a single fan stage from the fan frame of reference.

Shifting to a rotating frame of reference must be accomplished for the entire flowfield. It is apparent that from the blade point of reference the duct wall upstream of the fan appears to be rotating in the opposite direction. The WIND code has a moving wall option that can be specified as rotational motion about an axis. Therefore by “spinning” the wall, the wall boundary condition is transformed into the rotating frame.

Finally, from the blade point of view, the inlet flow into the duct upstream appears to be spinning with a solid body rotation. The WIND code contains an option for specifying a rotating inlet flow. Therefore, by specifying both an inlet flow rotation and a wall rotation in a direction opposite to the rotation of the fan, it should be possible to

perform a steady state rotating solution of the flow through an axially symmetric duct into the first stage of a turbo-fan engine.

4.3. Rotating Frame Test Case

Before attempting to run the large fan model some simplified test cases were generated to verify the “as-advertised” operation of the WIND code in the rotating frame of reference. A simple cylinder model was developed as shown in Figure 98 and Figure 99. The diameter of the cylinder was chosen as 50 inches. The maximum and minimum circumferential nodes are joined at a coupled boundary visible as a axial plane in Figure 98. The solution was run for an axial Mach number of 0.5 for both an inertial frame of reference calculation and a rotating frame of reference calculation. Steady inflow and outflow boundary conditions were applied. Flow was assumed to be inviscid with walls. The rate of rotation for the rotating frame of reference was 1500 radians/sec. The solution is obviously a uniform flow with no gradients in the inertial frame.

Comparison of the inertial and rotating solution revealed difficulties with the rotating frame of reference solution. For the rotating frame solution, there were several discrepancies. Figure 100 shows the Mach number solution on an axial plane at the midpoint in the duct using a rotating frame. The result of using an inertial solution to calculate the same Mach number condition is shown in Figure 101. The rotating frame solution shows several distortions. The Mach number decreases with radius and is distorted. There is also a distortion near the coupled boundary seen as a straight line

extending vertically from the axis. This is in comparison the very flat Mach number profile shown in Figure 101.

A more detailed comparison is shown in Figures 102 and 103. Figure 102 compares the solution along the vertical radius extending downward from the centerline axis. This radius was selected because it is opposite from the coupled boundary and therefore should show the least effect from the error that seems to be present at the boundary. While the solution in the inertial plane is a flat line as it should be, the solution in the rotating frame is seen to decrease with radius in a generally linear fashion. The solution along a circumferential line is shown in Figure 103. Once again, the inertial solution is seen to be essentially flat while the rotating solution varies. The solution appears to be roughly periodic. Also note the apparent discontinuity near the endpoints, which are located at the coupled boundary.

The static pressure results shown as pressure contour lines are shown in Figures 104-105. Since the ideal mass flow in the discharge coefficient calculation is very dependent on the static pressure, this variable is of great interest. As expected, the pressure field obtained in the rotating frame shows considerable variation in Figure 104. There is a strong gradient at the coupled boundary and there appears to be somewhat of a radial pressure gradient. Figure 106 shows that along a vertical radius, the pressure increases with radius and is discontinuous at the centerline. As shown in Figure 107, the pressure has discontinuous jumps at the coupled boundaries and varies along the circumference

One final look at the rotating frame results is shown in Figure 108. In this figure the velocity vectors in the y-z plane are shown for the interior points. Clearly, the flow contains spurious velocity components and gradients and is not everywhere zero as a proper solution would be for axial flow. The interesting thing to note is that in general there seems to be a clockwise rotation of the flow.

It is clear from these results that there were several problems present in the WIND code implementation of a rotating frame of reference. This led to an investigation of the causes of these errors and an effort to improve the WIND code performance. The specific difficulties identified and their solutions are discussed in the next section.

4.4. Corrections to the WIND Code Rotating Frame of Reference Capability

A number of changes and corrections were implemented in the WIND code to achieve a correct solution in the rotating frame. These changes can be grouped in to several categories. 1) Proper calculation of total conditions in a rotating frame, 2) proper extrapolation of velocities at boundaries in a rotating frame, 3) proper integration of scalar fluxes across control volume boundaries, and 4) proper calculation of the body forces present in a rotating frame.

4.4.1. Inlet Boundary Total Pressure and Total Temperature: Specification of inlet boundary conditions in the WIND code includes the specification of a total pressure and total temperature condition in the input file. Exercising the code revealed an error in

the proper calculation of total pressure and total temperature in the rotating frame of reference.

This error was the result of not taking into account the radial variation in velocity in the rotating frame. Total pressure and total temperature are functions of the reference frame.

$$p_t = p(1 + \frac{\gamma-1}{2} M^2)^{\frac{\gamma}{\gamma-1}} \quad \text{Eq. 41}$$

$$T_t = T(1 + \frac{\gamma-1}{2} M^2) \quad \text{Eq. 42}$$

In the inertial frame the Mach number

$$M_I = \frac{|\vec{v}|}{c} \quad \text{Eq. 43}$$

is constant, but in the rotational frame

$$M_R = \frac{|\vec{v} - \vec{u}|}{c} = \frac{|\vec{w}|}{c} \quad \text{Eq. 44}$$

and Mach number varies with radius since $\vec{u} = \vec{\omega} \times \vec{r}$. Since the frame velocity $\vec{u} = \vec{\omega} \times \vec{r}$ is different for every radial location $\vec{r}$, the total pressure and total temperature in the rotating frame also vary with radius. Therefore fixing the total pressure and total temperature in the inertial frame at a constant value, leads to a radial variation in the rotating frame.

A correction was made in the WIND code to provide correct inlet conditions. The correct inlet static conditions are first calculated in the inertial frame to determine the required total temperature and total pressure. Then the frame velocity is added to obtain

the correct values for Mach number in the rotating frame, M_R and the corresponding total pressure and temperature.

4.4.2. Inlet Characteristics for Rotational Frame: Rotational velocities must also be taken into account when calculating the inlet boundary values using characteristics equations. In the WIND code for subsonic inflow, the normal inlet velocity at the boundary is calculated by use of the adjacent characteristic values, specifically the third characteristic [34]:

$$C_3 = U_n - \frac{2}{\gamma - 1} c \quad \text{Eq. 45}$$

When total temperature is specified to remain constant this results in a set of two equations and two unknowns (U_n and T) by expressing the speed of sound in terms of static temperature.

$$C_3 = U_n - \frac{2}{\gamma - 1} \sqrt{\gamma R T} \quad \text{Eq. 46}$$

$$T = T_t \left(1 + \frac{\gamma - 1}{2} M^2 \right)^{-1} = T_t \left(1 + \frac{\gamma - 1}{2} \frac{U_n^2}{\gamma R T} \right)^{-1} \quad \text{Eq. 47}$$

For the case where there is a significant velocity parallel to the inlet boundary the expression for temperature becomes:

$$T = T_t \left(1 + \frac{\gamma - 1}{2} M^2 \right)^{-1} = T_t \left(1 + \frac{\gamma - 1}{2} \frac{(U_n + U_p)^2}{\gamma R T} \right)^{-1} \quad \text{Eq. 48}$$

For the case of interest, flow in an axially symmetric duct with a rotating frame of reference, the frame velocity is parallel to an inlet plane normal to the axis. In this case

$$U_p = \sqrt{u_y^2 + u_z^2} \quad \text{Eq. 49}$$

$$u_y = -\omega_x r_z \quad \text{Eq. 50}$$

$$u_z = \omega_x r_y \quad \text{Eq. 51}$$

Therefore, there must be a correction applied at every point on the boundary to account for the variation in total parallel velocity component. This correction was applied to the WIND code.

4.4.3. Wall Boundary Pressure: At wall boundaries, a “correction” was made in the WIND code to the static pressure for rotation. This was unnecessary. While total pressure is frame dependent, static pressure is independent of the frame of reference. This “correction” for static pressure was removed.

4.4.4. Wall Velocity for Slip Wall: At a slip wall boundary, the wall velocity is determined by an extrapolation from the adjacent point. In the simplest case, the velocity at the boundary is set equal to the velocity parallel to the boundary at the first grid point adjacent to the boundary. However, this result is not correct in a rotating frame. It should be formulated to take into account the motion of the frame of reference. This can be seen by examining the transform

$$\vec{v}_{bound} = \vec{v}_{adj} \quad \text{Eq. 52}$$

$$\vec{w}_{bound} = \vec{v}_{bound} - \vec{u}_{bound} = \vec{v}_{bound} - \vec{\omega} \times \vec{r}_{bound} \quad \text{Eq. 53}$$

$$\vec{w}_{adj} = \vec{v}_{adj} - \vec{u}_{adj} = \vec{v}_{adj} - \vec{\omega} \times \vec{r}_{adj} \quad \text{Eq. 54}$$

Therefore in the rotating frame:

$$\vec{w}_{bound} = \vec{w}_{adj} + \vec{\omega} \times \vec{r}_{adj} - \vec{\omega} \times \vec{r}_{bound} \quad \text{Eq. 55}$$

Extrapolation from the adjacent point to the boundary requires a correction based on the radius of each point. This correction was implemented in the WIND code to give correct results on slip walls in a rotating frame of reference.

The effect of the error is shown in Figure 109 and Figure 110. Figure 109 shows the uncorrected velocity vector component in the rotating frame at a slip boundary. Careful examination of the vector at the wall shows it is equal to the adjacent vector. The error can be seen more clearly by transforming back into the inertial frame as shown in Figure 110. Figure 111 and Figure 112 shows the proper result after the correction is applied.

4.4.5. Coupled Boundary: In the WIND code abutting zone boundaries are coupled together through a Roe [34] coupling process. In performing the rotating fan calculation, the solution for a single blade row is calculated by applying periodic boundary conditions, implemented as shown in Figure 113. Running the WIND code for this configuration revealed a difficulty on the coupled boundary.

To illustrate this problem, the cylindrical grid shown in Figure 113 was run for one iteration. Recall that all calculations are done in Cartesian coordinates. The coupled boundary is shown highlighted. Flow was initialized with a Mach number of 0.1 along the axis of the cylinder, the positive x-axis. There is no crossflow velocity in the yz plane. Figure 114 shows the velocity in the rotating frame of reference. The rotational motion of the flow in this frame is evident.

Figure 115 shows the velocity vector in the inertial frame at the uncorrected coupled boundary after 1 iteration. Since the flow is only axial, there should be no

crossflow in the yz plane, yet it is evident that an additional velocity has been introduced all along the coupled boundary.

This difficulty is the result of an extrapolation error. Figures 116-118 show the result of extrapolating the rotational velocity according to a 2nd order fully upwind method in computational space as implemented in the WIND code. The solid lines represent the correct values of w_z in Figure 116 and w_y in Figure 117 and Figure 118 for grid points along a radius of 1 ft and a rotational rate of 1000 radians/sec. The extrapolation calculated by the WIND code is indicated by the dashed lines. The extrapolation from either direction is based on the two points adjacent to the coupled boundary. The figures show an extrapolation assuming a coupled boundary at the 10th node

The extrapolated values do not match the actual value at the coupled boundary. This is the case even for a calculation in the inertial frame. However in the inertial frame, the slopes are much smaller and as a result the corresponding error in performing the extrapolation is much smaller. Indeed for the case specified, all cross velocities are zero, there are no slope differences and the extrapolation should yield an exact solution of zero. A significant error has been introduced merely by switching to the rotating frame. The rotation of the frame has the effect of introducing a significant radial gradient into the flow, which is present at all points in the flow field. This gradient is equal to the rotation rate.

$$\frac{dw_z}{dy} = \frac{dv_z}{dy} + \omega_x \quad \text{Eq. 56}$$

$$\frac{dw_y}{dz} = \frac{dv_y}{dz} - \omega_x \quad \text{Eq. 57}$$

The larger the rotation rate, the larger the curvature of the velocities in Figures 116 and 117 and the larger the corresponding extrapolation error. As the rotation rate is decreased, the curves become flatter and the corresponding error decreases.

The easiest way to correct this is to avoid it altogether. Modification was made to the WIND code to perform all coupled boundary calculations in the inertial frame. This involved transforming the appropriate values back into the inertial frame using the transforms given previously, performing the appropriate extrapolation, and then transforming the extrapolation back to the rotating frame.

The result of this correction is shown in Figure 119. Values of v_z along a given circumferential line are shown with and without the correction. The end points are coupled to each other. Without correction there is a clear error, with the correction the line remains flat.

4.4.6. Singular Axis Corrections: One of the boundary conditions available in the WIND code is a singular axis. On the axis the values are set by averaging the adjacent values at the nodes shown in Figure 120. Direct averaging of the values at $j = 2$ is not appropriate for a rotating frame. If a flow field that is zero everywhere is considered, then in the rotating frame the velocity would be equal to just the frame velocity, $\vec{u} = \vec{\omega} \times \vec{r}$. The correct value at the singular axis should be velocities of zero if the rotation is centered on the singular axis. Obviously at $j = 2$, the velocities are greater than zero and

the average most likely will not equal zero. Also consider that the frame velocity is a linear function of radius and therefore should not be averaged.

There are two solutions to this error. For the case where the singular axis coincides with the axis of rotation and a symmetrical flow solution exists, the cross flow velocities, w_y and w_z can simply be set to zero. Indeed the WIND code contains options for this setting even in the inertial frame. In the general case however, the flow solution may not be symmetric about a center axis. The more general solution therefore is to perform the averaging just on the inertial frame components.

An identical problem arises with the relaxation option available in the WIND code for the singular axis boundary condition. Referring to Figure 120, the relaxation method calculates the values at the singular axis ($j=1$) as an average of the values at $j=3$. Then for each k , values at $j=2$ are calculated by averaging the values at $j=1$ and $j=3$.

To avoid averaging errors, all values involved in the averaging calculation should be transformed to the inertial frame, calculations performed, and then results transformed back to the rotating frame of reference. These corrections were made to the WIND code.

4.4.7. Corrections Involving Body Forces: The default explicit solver in the WIND code is a Roe's second order flux-difference splitting algorithm, modified for stretched grids. This method is based on a control volume approach, with fluxes integrated across the volume boundaries. Because of the body force terms and the velocity of the frame, modifications must be made to perform these integrations correctly.

Figure 121 and Figure 122 show the cross velocities w_y and w_z on a constant x plane after one flow field iteration. The flow was initialized as constant axial flow with no cross velocities, yet after only 1 iteration, cross velocities begin to appear as shown in the figures. Also visible in the figure is the boundary coupling error discussed above.

The finite volume control volumes implemented in the WIND code are shown in Figure 123. For simplification, only two dimensions are shown. The volume boundaries in the WIND code are defined as half way between the grid points. For an evenly spaced grid, this results in the grid point being located at the centroid of the volume. However, for a stretched grid, this is not the case. As shown in Figure 123 the grid point and the centroid are at two different locations. This is significant because of the presence of body forces in the rotating frame that are not present in the inertial frame.

The body forces in the rotating frame are the centrifugal acceleration and the Coriolis acceleration which appear on the right-hand side of the momentum equation. The centrifugal acceleration term appears as $\rho[-\vec{\omega} \times (\vec{\omega} \times \vec{r})]$. Since the rotation rate is constant for the case under consideration, this term is primarily a function of radius. The Coriolis acceleration is expressed as $\rho[-2\vec{\omega} \times \vec{w}]$. For a constant rotation rate, the velocity in the rotating frame is the primary variable.

The proper application of these accelerations must be determined for the control volume used in the solution. Consider first the centrifugal acceleration. The total centrifugal force on the control volume can be expressed as

$$\vec{F}_{centrifugal} = \iiint_V \rho[-\vec{\omega} \times (\vec{\omega} \times \vec{r})] dx dy dz \quad \text{Eq. 58}$$

The components can be separated to give

$$\vec{F}_{centrifugal} = \iiint_V \rho [-\vec{\omega} \times (\vec{\omega} \times \vec{r}_x) - \vec{\omega} \times (\vec{\omega} \times \vec{r}_y) - \vec{\omega} \times (\vec{\omega} \times \vec{r}_z)] dx dy dz \quad \text{Eq. 59}$$

or

$$\begin{aligned} \vec{F}_{centrifugal} = & \iiint_V \rho [-\vec{\omega} \times (\vec{\omega} \times \vec{r}_x)] dx dy dz + \iiint_V \rho [-\vec{\omega} \times (\vec{\omega} \times \vec{r}_y)] dx dy dz \\ & + \iiint_V \rho [-\vec{\omega} \times (\vec{\omega} \times \vec{r}_z)] dx dy dz \end{aligned} \quad \text{Eq. 60}$$

Expanding the acceleration term of the first volume integral:

$$\vec{\omega} \times (\vec{\omega} \times \vec{r}_x) = (-\omega_y^2 - \omega_z^2) r_x \mathbf{i} + \omega_x \omega_y r_x \mathbf{j} + \omega_x \omega_z r_x \mathbf{k} \quad \text{Eq. 61}$$

Assuming a constant rotation rate the first volume integral becomes:

$$[(-\omega_y^2 - \omega_z^2) \mathbf{i} + \omega_x \omega_y \mathbf{j} + \omega_x \omega_z \mathbf{k}] \iiint_V \rho r_x dx dy dz \quad \text{Eq. 62}$$

The x-coordinate for the center of gravity of the control volume is:

$$\bar{r}_x = \frac{1}{M} \iiint_V \rho r_x dx dy dz \quad \text{Eq. 63}$$

where M is the mass contained in the control volume.

Assuming a constant density in the control volume, the mass of the control volume is $M_{cv} = \rho V$. By substitution then, the integrand of the first volume integral can be expressed as

$$[(-\omega_y^2 - \omega_z^2) \mathbf{i} + \omega_x \omega_y \mathbf{j} + \omega_x \omega_z \mathbf{k}] \rho V \bar{r}_x = \rho V [\vec{\omega} \times (\vec{\omega} \times \vec{r}_x)] \quad \text{Eq. 64}$$

By symmetry, the other components can also be evaluated and therefore the centrifugal force on a control volume can be expressed as

$$\vec{F}_{centrifugal} = \rho V (\vec{\omega} \times (\vec{\omega} \times \vec{r})) \quad \text{Eq. 65}$$

where density is assumed constant in the control volume, V is the volume of the control volume and $\bar{\vec{r}}$ is the centroid or center of mass of the control volume.

As originally implemented the WIND code applies the centrifugal force at the location of the node instead of the center of mass. This results in an error proportional to the difference between radius of the node and the radius of the centroid. For an evenly spaced grid the node and the center of mass coincide and there is no error, but the more the grid is stretched, the more error is introduced. The error is also proportional to the rotational rate. Doubling the rotation rate doubles the magnitude of the error.

The Coriolis acceleration can also be examined in similar fashion assuming ρ and $\vec{\omega}$ are constant:

$$\begin{aligned}
 \vec{F}_{coriolis} &= \iiint_V \rho [-2(\vec{\omega} \times \vec{w})] dx dy dz \\
 &= -2\rho \vec{\omega} \times \frac{V}{V} \iiint_V \vec{w} dx dy dz \\
 &= -2\rho V (\vec{\omega} \times \bar{\vec{w}})
 \end{aligned}
 \tag{Eq. 66}$$

Therefore it is seen that the Coriolis acceleration should be calculated based on the average value of the velocity in the cell. It will be shown in the next section that a correction was required to calculate the average value on the solution vector in the cell.

4.4.8. Corrections Involving the Roe Method: A schematic of the basic Roe method in 1-dimension is shown in Figure 124. As previously mentioned the default explicit solver is Roe's second order flux-difference splitting algorithm, modified for stretched grids. The value inside of each control volume is assumed to be an average in the volume. At each iteration, fluxes at each boundary are evaluated and integrated to arrive at new average values in the control volume.

In the inertial frame the solution vector $\tilde{q}$ is represented by a single average value, inside each volume. The distribution of $\tilde{q}$ inside the volume is not specified and is unknown. The flux at each boundary is determined by Roe averaging [34] the left and right states at each boundary as shown in the figure. Various schemes exist for determining higher order accuracy of the left and right states at the boundary. However at each boundary the flux is assumed to be constant on the boundary.

In the rotational frame, it is known that there are rotational components that vary inside the control volume. This may impact the way some terms are evaluated. Consider first the value of the momentum terms, $\rho \tilde{w}$. In the control volume formulation, we assume this value to represent the average, $\overline{\rho w}$. The average can be expressed as:

$$\begin{aligned}
 \overline{\rho w} &= \frac{1}{V} \iiint_V \rho w^{\star} dx dy dz = \frac{1}{V} \iiint_V \rho (v^{\star} - u^{\star}) dx dy dz \\
 &= \frac{1}{V} \iiint_V \rho v^{\star} dx dy dz - \frac{1}{V} \iiint_V \rho (\omega \times r)^{\star} dx dy dz \\
 &= \overline{\rho v} - \rho \left(\omega^{\star} \times \frac{1}{V} \iiint_V r^{\star} dx dy dz \right) \\
 &= \overline{\rho v} - \rho (\omega^{\star} \times \bar{r}) = \overline{\rho v} - \rho u_{centroid}^{\star}
 \end{aligned} \tag{Eq. 67}$$

This expression illustrates the fact that the average value of the momentum terms in the rotating frame are equal to the average value in the inertial frame plus the rotational term evaluated at the center of mass of the control volume. Therefore, in any calculations involving the use of the momentum terms, care must be taken when calculating in the rotating frame that the momentum terms are evaluated at the center of mass. As has already been discussed, the current implementation of the WIND code evaluates terms

based on the node location, which is not coincident with the center of mass in a nonuniform grid.

Similar results can be shown for the energy term, $\overline{\rho E^*}$, assuming that the velocity in the inertial frame, $\vec{v}$, is constant in the control volume:

$$\begin{aligned}
 \overline{\rho E^*} &= \frac{1}{V} \iiint_V \rho E^* dx dy dz = \frac{1}{V} \iiint_V \rho (E - \vec{u} \cdot \vec{v}) dx dy dz \\
 &= \overline{\rho E} - \rho \vec{v} \cdot \frac{1}{V} \iiint_V (\vec{\omega} \times \vec{r}) dx dy dz \\
 &= \overline{\rho E} - \rho \vec{v} \cdot \left(\vec{\omega} \times \frac{1}{V} \iiint_V \vec{r} dx dy dz \right) \\
 &= \overline{\rho E} - \rho \vec{v} \cdot (\vec{\omega} \times \vec{r}) \\
 &= \overline{\rho E} - \rho \vec{v} \cdot \vec{u}_{centroid}
 \end{aligned}
 \tag{Eq. 68}$$

The terms in the continuity equation remain unchanged since density is independent of the frame of reference.

A similar issue exists at the flux boundaries. By the same method, it can be shown that the average value of the boundary states, $\vec{q}_L$ and $\vec{q}_R$, on the cell boundaries is equivalent to the value evaluated at the centroid of each cell face.

Changes were implemented in the WIND code to correct for these effects. This involved adding calculations for the centroid of each control volume cell and the centroids of each control volume face. This required additional computer memory storage. For a given grid with n nodes, the WIND code stores coordinate data for each node. The addition of the centroid calculations requires a centroid coordinate for each cell and three of the cell faces. This represents an increase in geometry data storage of 400

percent. However, for the cases examined this represented about a 33 percent increase in total code storage.

The default methods in the WIND code correct for grid stretching. This correction is based on the geometry of the node coordinates. Since the centroid locations are not the same as the cell nodes, using centroid locations would require modification of the grid stretching corrections. Rather than modifying various schemes in the WIND code, the calculations for the boundary states were performed in the inertial frame and then transformed by the appropriate relations back into the rotating frame.

4.4.9. Turbulence Model Corrections: The WIND code has several turbulence models available as discussed earlier in Section 3.5.2. One key parameter is the turbulent viscosity. The Shear Stress Transport model uses flow vorticity to calculate the turbulent viscosity, but the addition of frame rotation affects the vorticity calculation since:

$$\begin{aligned}\vec{\Omega} &= \vec{\nabla} \times \vec{w} = \vec{\nabla} \times (\vec{v} - \vec{u}) \\ &= \vec{\nabla} \times \vec{v} - \vec{\nabla} \times (\vec{\omega} \times \vec{r}) \\ &= \vec{\nabla} \times \vec{v} - 2\vec{\omega}\end{aligned}\tag{Eq. 69}$$

The effect on the turbulence model then, is that the addition of a rotating frame results in the addition of turbulent viscosity everywhere in the flow. The solution to this was to input into the turbulence model the same vorticity it would receive in an inertial frame: $\vec{\nabla} \times \vec{v} = \vec{\nabla} \times \vec{w} + 2\vec{\omega}$. This resulted in a turbulent viscosity distribution with identical results regardless of rotation rate. This correction was implemented in the WIND code.

4.4.10. Speed of Sound Corrections: Throughout the WIND code, the speed of sound is often calculated as a function of the fifth element of the solution vector, ρE .

However when calculating in the rotating frame, the fifth element of the solution vector is no longer ρE but rather it becomes ρE^* . Great care must be taken to properly calculate the speed of sound using the correct formulation:

$$c = \sqrt{\gamma R T} = \sqrt{\frac{\gamma R (\rho E)}{\rho c_v}} \quad \text{Eq. 70}$$

for the inertial frame, but

$$c = \sqrt{\gamma R T} = \sqrt{\frac{\gamma R (\rho E^* + \rho \mathbf{u} \cdot \mathbf{v})}{\rho c_v}} \quad \text{Eq. 71}$$

for the rotating frame.

Note that the speed of sound is also a function of radius in the term $\mathbf{u} \cdot \mathbf{v}$. There were several errors and omissions in the WIND code regarding the speed of sound calculation. The code was searched for such calculations and the appropriate corrections were made.

4.5. Demonstration of Rotating Frame Corrections

To verify the proper operation of the WIND code in the rotating frame of reference, two test cases were developed and run. The first case was the inviscid cylinder case discussed earlier in section 4.3. This case was run again with the corrected code. The Mach number solution along a radial and circumferential line is shown in Figures 125 and 126 in comparison with the previous incorrect solution and a nonrotating solution. Although the new solution is not perfectly constant as should be the case for a perfect solution, it is obviously much improved. Similar results are shown for pressure in Figures 127 and 128.

The second test case was to calculate discharge coefficients in a bellmouth flow meter. This requires 3-D viscous flow in a rotating frame. The grid developed for this test case is shown in Figure 129. It is a 3-D wedge of angle 15 degrees. The profile of the bellmouth is identical to the profile of the experimental bellmouth as shown in Figure 23. To decrease solution time, the grid was split into eight zones by use of a WIND utility program and calculated using eight processors. The results were recombined after the run was completed.

The inlet flow was set to hold the total conditions and an outlet static pressure was matched to the inlet total pressure to provide a Mach number of 0.5 in the bellmouth throat area. The centerline of the bellmouth was set to a singular axis condition and the walls were viscous. The radial boundaries were coupled to provide periodic boundaries. The calculation assumed viscous flow with the SST turbulence model selected.

Comparisons of the inertial and rotating frame calculations are shown in Figures 131-140. Overall, the agreement is seen to be quite good. Figure 130 shows the plane of the grid on which the comparison is made.

Two centerline values are shown in Figures 131 and 132. The centerline velocity is shown in Figure 131. There is some disagreement, about 5% in the values, in the upstream portion of the duct where velocities are low. The difference quickly decreases to much less than 1% as the bellmouth wall converges and the velocities increase. However when examining the centerline pressure it is seen in Figure 132 that the differences are very small even in the upstream portion of the duct. The percent difference between the rotating and inertial solution in this portion of the duct is on the

order of 0.05% difference. This small difference in pressure compared to a larger difference in velocity is consistent with the earlier mentioned principle that flow rate is very sensitive to pressure at lower Mach numbers.

Visible in the plots are discontinuities in the trace marking the percent difference between the two solutions. These discontinuities occur at the grid zone boundaries where the solution is affected by coupled boundary treatment.

Of critical interest for evaluating the discharge coefficient is the wall pressure. The wall pressure of the two solutions is compared in Figure 133. Agreement is excellent with the wall pressure matching to within 0.03 percent difference and even 0.015% difference for most of the bellmouth surface.

Comparisons of radial profiles at Station 8 are shown in Figures 134-135. The axial velocity component is shown in Figure 134. There is some discrepancy seen near the wall in the boundary layer and a small difference near the singular axis. The variations in these regions reach a maximum of about 0.5%

The other two velocity components are shown in Figures 135 and 136. Because these components are very small, the percent difference is not shown. It is useful to note the axial velocity at this station is about 550 ft/s. The y-component of velocity, shown in Figure 135 is seen to agree within about 0.05 ft/s. The y-component in this plane is primarily a radial component. The z-component is shown in Figure 136. Values agree to within a value of about 0.9 ft/sec. There is a noticeable jump in the rotating frame at the singular axis. Also there is a marked peak in the boundary layer region for the rotating frame solution. The z-component in this plot is primarily a circumferential velocity.

The radial pressure profile is shown in Figure 137. Agreement is seen to be very good with some difference near the centerline. In this area the rotating frame solution is about 0.01 percent larger than the inertial frame solution. In both solutions, the pressure is very flat at Station 8. The static temperature exhibits similar good results as shown in Figure 138.

To provide a general indication of the agreement in other regions of the flowfield, the static pressure and temperature results are compared along the $j=25$ gridline which is highlighted in previous Figure 130. The pressure result is shown in Figure 139. The difference between the two solutions is seen to be less than about 0.02 percent difference for most points, with a maximum percent difference of about 0.035. Temperature results are very good also, within 0.006 percent difference as shown in Figure 140.

Overall the agreement is seen to be quite good, and the results were evaluated as being sufficient to continue with the problem of interest. Suggestions for improving the remaining differences will be made in Conclusions and Recommendations Section of this dissertation.

4.6. Fan Selection

Demonstration of the capability to calculate the effect of a rotating fan on the discharge coefficient of an inlet bellmouth requires the definition of a blade geometry. The blade geometry of engines tested at altitude facilities is usually very complex and is

typically proprietary information of the engine manufacturer. Therefore, acquiring these geometries and the accompanying data is usually difficult.

In consideration of this, attention was turned to other sources of blade geometry and data. Rotor 67 [38] is a well defined compressor stage, designed and tested to provide data for comparison to CFD solutions. Rotor 67 and its performance are described in detail in the reference. Based on the availability of the data and the public sourcing, Rotor 67 was chosen as a baseline geometry.

Rotor 67 is shown in Figure 141. Rotor 67 is a low-aspect-ratio transonic fan rotor. The relative Mach number at the fan tip was 1.38. In the research described in Ref [38] laser anemometer surveys for operating conditions near peak efficiency and stall were made of the flowfield. The fan was operated at a design rotational speed of 16,043 rpm or 1680 radians/sec.

Figure 142 shows the pressure ratio performance for Rotor 67. At peak efficiency it operates near a pressure ratio of 1.65, with a maximum flow rate of 34.96 kg/s (77.07 lbm/s). Shown in Figure 143 are the experimentally measured relative Mach number contours at various spans for peak efficiency and near stall flow.

4.7. Computational Grids

The original configuration of Rotor 67 as tested by NASA contained an annular inlet. This differs from the typical turbofan inlet. The inlet geometry was modified to include a spherical bullet nose, typical of a turbofan inlet. This configuration is shown in

Figures 144-146. The bullet nose was made tangent to the fan hub at the point of attachment.

Two grids were developed for the compressor and associated ducting. The first grid consisted of the compressor installed in a straight duct. The grid is shown in Figures 147-149. Figure 147 shows a view of the entire grid. Every other grid line has been removed from the figure for clarity. The entire fan is modeled as one blade out of 22 with periodic boundary conditions applied at the sides of the wedge, therefore the grid describes a wedge of 16.36 degrees.

A closer examination of the grid in the region of the blade is shown in Figure 148. Details of the blade itself have been left out for clarity in the figure. Figure 149 shows a side view of the grid. A clearer view of the grid around the blade is shown in the top view as illustrated in Figure 150. The grid is tightly packed near the blade surface. The boundary condition on the blade surface was specified as a no-slip condition. Note that the wedge “shifts” to put the symmetry line at the mid-passage between adjoining blades. However, at each point on the boundary the corresponding match point is precisely rotated 16.3636 degrees and coupled, therefore the periodic boundary conditions hold. The tip of the blade is assumed to go all the way to the wall. The tip clearance is zero.

The grid was split into 14 zones for multiple CPU processing. These zones are shown in the wire frame diagram in Figure 151. There are six zones clustered around the fan blade. The size and location of each grid was chosen to try to keep the number of grid points as nearly equal as possible in each zone while also keeping zone boundaries away from difficult flow features. The grid is structured with a total of 638,305 grid points.

The second grid is shown in Figures 152 and 153. It is similar to the first grid except the engine inlet has been fitted with a bellmouth flow meter. The bellmouth profile is of the same shape as the experimental bellmouth modeled in the previous sections, as shown in Figure 23. The grid is split into 15 computational zones and the number of total grid points is 777,955. The compressor face is located approximately 2/3 diameters downstream of the bellmouth throat or tangent point.

4.8. Computational Results of Fan Installed in Straight Duct

A solution for the fan installed behind a section of straight duct was run using the first grid as shown in Figure 147. The inlet conditions were set to hold inlet total pressure at 14.696 psia and total temperature at 518.67 R in the fixed frame. This matches the inlet total condition of the experiment described in [38]. The outlet condition was set by specifying an outlet static pressure. The outlet pressure will establish a pressure ratio across the fan. Some trial-and-error was required to arrive at a satisfactory outlet pressure. The goal was to set the static pressure at the outlet downstream of the fan such that the total pressure ratio and mass flow would be close to the peak efficiency point shown on the performance curve in Figure 142. Through this method the outlet static pressure was chosen to be 15.8 psia.

Upstream of the fan the walls were assumed to be viscous. The walls are stationary in the inertial frame but are set to rotate with the frame in the rotating frame

solution. In the region around the tip and the hub of the blade the wall condition was assumed to be inviscid.

Some difficulty was encountered in trying to “start” the solution. The default behavior of the WIND code is to initialize each zone to a fixed velocity in the frame of reference selected. The only exception to this is that a solid body rotation can be specified for a zone with an inlet flow boundary. The result was that upon startup the entire flow field was initialized with a flow that was a constant axial flow in the rotating frame. This translated into a rotating flow in the duct in the inertial frame, which is incorrect. This difficulty was overcome by specifying each zone to have an inlet flow boundary for one iteration and specifying a solid body rotation for each zone. This resulted in a solution file in which the flow had a constant axial velocity down the duct in the inertial frame. The modified boundary conditions were then set back to their proper boundary conditions and the solution restarted.

Even after this treatment, the startup condition was not suitable for the blades. The fan is not “started” and in essence the flow was stalled over the blade. Attempts to solve the flow in this situation were unsuccessful. Disturbances from the blade stall propagated upstream and interfered with the inlet boundary condition causing instability. The solution was finally started by “freezing” the flow in zones upstream of the fan. This provided the fan with a constant rotating inlet flow until the solution started around the blades.

The WIND code contains a TVD (Total Variation Diminishing) option to help in overcoming strong startup transients. This capability was used until the solution had

steadied and then was turned off in most of the flowfield. TVD can be enabled on a zone by zone basis. To help overcome some instabilities that were developing near the center of the blade, the TVD option was left enabled in the region around the blade. Various CFL numbers were tried. In general, stability was difficult to maintain with CFL greater than one. For the results presented the CFL was set to about 0.25. Convergence required approximately 300,000 iterations.

4.8.1. Comparison of Results with Experiment: Computed Mach number profiles around the blade are shown in Figures 154-156 for various stations along the blade span. The Mach contours at a span location of 10% span from the tip are shown in Figure 154. The experimental results for the same location are seen in Figure 143a. The experimental results are presented for two performance points, near peak efficiency and near stall flow. It is interesting to note that for the shock location in the experiment, the Mach number seems to decrease gradually behind the shock. Since a shock “fan” is not possible, it is likely that this is an artifact of the experimental data technique. The Mach contours presented are an average based on laser anemometer measurements of each blade passage. Since each blade is likely to be slightly different, the shock will not be at exactly the same location. In addition, in the real experiment the shock is likely to shift position somewhat with time. Also, the laser anemometer system makes use of seeding particles which will exhibit a lag relative to the fluid upon crossing the shock. The effect would be to smear the location of the shock, which seems consistent with the contours shown in Figure 143.

In the computational results an oblique shock is seen coming off the blade tip and impinging on the blade below. The approach Mach number at the tip is approximately 1.4. There is an immediate drop in Mach number behind the shock, as expected, then an acceleration back to around Mach one. A small wake appears behind the trailing edge. In general, there is good agreement of the flow structure in the computational result and in the experimental results for near peak efficiency flow as shown in Figure 143a.

Results calculated for 30% span from the blade tip are shown in Figure 155. Since this location is at a smaller radius, the rotational velocity is smaller and therefore the approach Mach number is about 1.25 at the leading edge. The shock angle is closer to 90 degrees. Comparison of Figure 155 and 143b show good agreement.

Predicted Mach number contours at 70 percent span from the shroud are shown in Figure 156. At this radius the approaching flow is subsonic with a value of about 0.95. There is a small supersonic bubble located just behind the blade tip. This agrees with the experimental results shown in Figure 143c.

In order to characterize the performance of Rotor 67, two measurement stations were set up to obtain aerodynamic performance data. The first was located at 1.82 inches in front of the leading edge at the tip, the other 3.485 inches behind. At each measurement station a total pressure, total temperature, static pressure and flow angularity were measured and tabulated near the stall and peak efficiency operating points.

Comparison to the upstream data is shown in Figures 157-159. Comparison of total pressure in the inertial frame is shown in Figure 157. The total pressure profile at the

outer radius seems to indicate that the experimental data has a thicker boundary layer than the computational results. This may be the result of the annular inlet passage to the rotor in the experiment. At the root of the blade, the computational solution does not decrease. This is because of the spinner which is present in the computational results but not in the experiment. The spinner rotation imparts motion to the boundary layer flow, increasing the total pressure.

Total temperature in the inertial frame is compared in Figure 158. It is clear that for both the computation and the experiment, the flow is adiabatic and the total temperature is virtually undisturbed.

There are several features to note in the static pressure comparison shown in Figure 159. First, for the case being computed, the addition of the engine spinner means that the flow area of the computational case is larger at the measurement station than the flow area during the experiment. This is manifested in Figure 159 as a somewhat higher pressure across the plane. The computational results shows lower pressure near the hub. This is due to the additional velocity induced by the rotating hub. By comparison the experimental data shows a noticeable pressure gradient across the radius.

4.8.2. Examination of Results in a Straight Duct: Since the purpose of this calculation is to determine the discharge coefficient in a bellmouth upstream of the engine, the flow field upstream of the fan is of primary interest even though it is a constant area duct section. A number of contour plots are presented to examine this area.

The Mach number profile in the inertial frame is shown in Figure 160. The Mach number approaching the fan at the upstream end of the duct is about 0.45. It is seen that

the Mach number is very uniform approaching the fan. The flow begins to slow in the center of the duct as it approaches the engine spinner. As the flow progresses past the spinner, Mach number contours appear radially across the duct. These contours are the result of the pressure disturbances caused by the fan blades passing through the flow. These disturbances propagate upstream, causing distortions in Mach number. The scaling of the Mach contours does not permit the visualization of fine details in this plot, but more detail will be seen in the figures to follow.

It can also be seen that at the centerline of the duct there is a distortion of the Mach contour. This distortion is due to numerical difficulties at the centerline. The distortion is also visible in several remaining figures. Attempts to remove this distortion were only partially successful. However, the judgment was made that the results are still useful. The distortion is only about .7% of the Mach number. Furthermore, the distortion is located at the centerline, far away from the wall pressure location, which is the area of interest. Comments about the cause of this distortion and some possible remedies will be reserved for the Conclusions and Recommendations Section.

The axial or x-component of velocity in the inertial frame is displayed in Figure 161. As expected, this plot is similar to the Mach number. Figure 162 shows the y-component of velocity. In the orientation of the plane shown, the y-component represents the velocity in the circumferential direction. Careful examination of the fluctuations along the wall reveals an interesting phenomenon. As the disturbances progress upstream, there appear at first two “cells” stacked on top of each other, then a series of single cells,

displaced downward from the wall, and then a series of two cells once again. The origin of this phenomenon will be more clear in examination of some pressure plots to come.

The third velocity component, the z-component, is displayed in Figure 163. This component represents the radial component on the plane shown. The most notable feature is the flow turning over the engine spinner to enter the fan. These contours are disturbed also near the wall.

The static temperature contours are shown in Figure 164. These contours are well defined. The variations are not very large and result from the pressure disturbances. Very visible in this plot is the alternating pattern of the disturbances, showing up here as a series of vertical cells interspersed by a “y” shaped region.

Finally, the static pressure contours are examined in Figures 165 – 169. A general view of the pressure contours is seen in Figure 165. Pressure disturbances are seen to propagate upstream in a well defined pattern along the wall. The static pressure on the wall surface in the region of the fan is shown in Figure 166. from a point of view looking down the blade span. As can be seen the pressure disturbances arise as shocks off the leading edge of the fan blades as they sweep through the flow. Detail of the area near the tips is shown in Figure 167.

Figures 166 - 169 as shown represent a view of the flow from the blade frame of reference, or the rotating frame of reference. Alternately, the figures may be seen as an instantaneous snapshot of the view in the inertial frame. As the blade passes through the flow, it traces out a disturbances that propagates upstream forming initially a helix pattern. The effect of this propagation is seen more clearly in a complete three-

dimensional view of the wall shown in Figure 168. The angle formed by the advancing disturbances is set by the speed of the fan blade through the flow and also the speed of upstream propagation. Since the speed of the leading edge is a function of radius, and the flow area is changing over the spinner, the angle and speed at which the disturbance leaves the blade and propagates through the flow can vary at different locations upstream of the blade. The range of Mach number approaching the leading edge along the span varies from 0.75 at the hub to 1.4 at the tip.

It is also apparent in Figure 166 and Figure 168 that there is a change in the wall pressure pattern just upstream of the fan. The pressure pattern appears to diminish and then strengthen again. This feature of the flow also repeats as it moves upstream. This is seen in the pressure contours shown in Figure 169 where a series of pressure disturbances appear down the duct, punctuated by a flatter area separating different regions.

For insight into the mechanism of this behavior it is useful to examine the pressure contours on a series of successive axial planes shown in Figures 170-177. It appears that this phenomenon is caused by the tendency of the pressure disturbances to propagate outward while being constrained by the cylindrical duct.

Consider a single shock wave propagating from a single blade as shown in Figure 167. The disturbance initially propagates normal to the line of disturbance. This direction of propagation has both an upstream axial component of velocity, and a rotational component of velocity. It is this rotational component of velocity that should be considered when looking at Figures 170-177. In these figures the point of view is from

the fan blades looking upstream at successive slices of the pressure field as it propagates upstream, away from the blade.

In the initial view in Figure 170 a pressure distortion is seen in well defined peaks all along the circumference of the wall. This disturbance has a circumferential velocity of propagation. Since the temperature is fairly constant and flow velocities are similar in the region of the disturbance, it will tend to propagate at the same velocity. This holds true for the circumferential component as well. However, the inner disturbance away from the wall is at a different radius than the outer disturbance at the wall. Therefore, for the inner disturbance traveling at approximately the same velocity as the outer disturbance, the inner disturbance transverses a greater angular deflection than the outer disturbance because of the smaller radius. This results in the disturbance rotating at a higher angular velocity as the radius decreases. As can be seen in Figures 170-177 the disturbance begins to rotate from the shock angle corresponding to the initial Mach number until eventually it “splits” into two regions of high and low pressure as shown in Figure 173. The differential rotation continues until eventually, the high and low pressure regions in the different layer synchronize again and the two regions merge. This pattern continues up the duct and gives rise to the repeating pattern seen on the wall.

4.8.3. Discharge Coefficient in Straight Duct Upstream of Fan:

Characterization of the discharge coefficient is dependent upon the static pressure measurement made to determine the calculated ideal flow. Since there are considerable fluctuations of pressure on the wall, the method of determining the calculated ideal flow should be considered carefully.

A view of the static pressure on the wall along the axial direction is shown in Figure 179. The fluctuations due to the disturbances discussed previously is apparent. The view shown is from the blade frame of reference or the rotating frame. As such, it represents an instantaneous view of the pressure profile in the inertial frame.

The pressure distributions along the circumference of the wall at a constant axial location are shown in Figure 180 for a variety of different axial planes. This view subtends an angle of 16.3636 degrees or $1/22$ of the entire circumference, corresponding to one of the 22 blades in Rotor 67. This view is constant in the rotating frame of reference.

The distribution shown in the figure is moving however, in the inertial frame of reference. Therefore, the distribution can be viewed as a time trace of what a wall static pressure measurement would be exposed to if installed in the duct. Since there are a fixed number of blades rotating at a constant rate of rotation, the frequency can be easily calculated. For a rate of 1680 radians per second with 22 blades, a static pressure tap would see the fluctuation shown with a frequency of 5,882 Hz.

The most obvious means of establishing the discharge coefficient would be to calculate the ideal flow based on the average pressure at given axial location. This would require specifying a pressure measurement technique that could resolve the average of the pressure distribution in Figure 180 at the appropriate frequency. It seems likely that at these frequencies, if there is any significant length of tubing between the tap and the pressure sensor, the pressure measurement would tend to self average.

Assuming an appropriate measurement system, the average pressure can be calculated from the computations along the duct wall. Figure 181 shows the maximum, minimum, and average pressure at each circumference along the duct. As can be seen by the maximum and minimum values, the amplitude of the disturbance is cyclic along the duct. This is a result of the pressure disturbance propagation pattern discussed earlier. Interestingly however, the average value is very well behaved and smoothly increases to the static value of undisturbed flow away from the fan.

Using the average static wall pressure at each plane, the discharge coefficient along the duct wall was calculated and is shown in Figure 182. For a straight duct the discharge coefficient would be expected to gradually decrease as the displacement thickness increases. It is seen in Figure 182 that the discharge coefficient begins to decrease rapidly as it approaches the engine face. The decrease begins well upstream of the engine spinner nose, as indicated on the figure. An examination of the pressure contours in Figure 163 and the velocity contours in Figure 161 indicate why this is the case. The centerline flow begins to decelerate upstream of the spinner reducing mass flow in the center of the duct and accelerating flow near the duct wall, thereby lowering the static pressure and the resultant discharge coefficient.

4.9. Computational Results for Bellmouth Upstream of the Fan

Computations were also performed using the same flow conditions for a fan installed in a duct with a bellmouth as shown in the grid in Figure 151. A number of contour plots of the solution are presented in Figures 183 -189.

The first of these, Figure 183 shows the Mach number contour. The Mach number in the upstream portion entering the bellmouth is about 0.2. The Mach number in the throat is approximately 0.45. Figure 184 presents the corresponding profile of the x-component of velocity.

The contours of y-component of velocity are presented in Figure 185. As in the straight duct plots, this component in this plane is a circumferential component. It is apparent that in the throat region of the bellmouth, the presence of the bellmouth affects the flow distortions. In comparison to the straight duct, the distortions begin to dissipate rapidly as they feed back upstream into the expansion of the bellmouth. It is also interesting to note that the magnitude of the y-component is very small throughout the throat. Since this is the circumferential velocity, this indicates that the engine is imparting very little swirl velocity into the approaching flow.

Shown in Figure 186 are the contours of the z-component of velocity. This is a radial component in the plane shown, and therefore the flow turning inward at the bellmouth is plainly visible. The disturbances are visible near the wall as the flow enters the fan.

Temperature and pressure contours are shown in Figures 187 and 188. It is seen that the pressure disturbances dissipate rather smoothly in the bellmouth indicated by smooth contour lines. In the throat region, the disturbances upstream propagating pressure disturbances begin to be seen. Details of the pressure contours in this region are seen in Figure 189. In contrast to the straight duct, the pressure waves are significantly changed by the presence of the bellmouth expansion. They begin to damp quickly upstream of the engine spinner. The pressure wave pattern seen in the straight duct section also does not establish as strongly.

The wall static pressure in the region of the throat just upstream of the fan is seen in Figure 190. It is clear in this view the pressure disturbances dissipate quickly upon passage upstream into the bellmouth expansion.

4.9.1. Discharge Coefficient of Bellmouth with Installed Fan: Once again, the instantaneous static pressure along the wall in the axial direction is shown in Figure 191. The fluctuations are seen to damp in the throat region of the bellmouth and smoothly decrease proceeding upstream along the bellmouth surface. Several circumferential instantaneous wall static pressure profiles are shown for various axial stations in Figure 192. As in the case of the straight duct, these profiles represent the temporal variation to which a wall pressure measurement would be exposed.

The maximum, minimum, and average pressure at each axial location is shown in Figure 193. In comparison to the similar results for the straight inlet duct section, there is a distinct maximum in the average value near the throat section. This maximum occurs about one half a diameter upstream of the fan. There is a relative minimum in static

pressure just upstream of the maximum value. This is a result of the locally reduced pressure in the region of the throat which was seen to be prevalent in all the bellmouth results examined in the previous sections with no engine installed.

The discharge coefficient based on average wall pressure is shown in Figure 194. Some oscillation of the discharge coefficient is noted in the upstream portion of this plot. This is due to small pressure oscillations on the wall in the curved region of the bellmouth and most likely indicates a need for finer grid resolution on the wall, since the wall pressure is very sensitive to slope discontinuities. However, these are very small and are hardly visible in the pressure plot as seen in Figure 191. The slight increase in the variation in the upstream direction is an indication of the increased sensitivity of discharge coefficient to pressure as the flow velocity decreases.

In the throat region, the discharge coefficient is seen to vary smoothly with distance, a maximum occurring just downstream of the throat and then to decrease quickly as the distance approaches the engine face.

It is of interest to compare the results with the fan installed and without the fan installed. Figure 195 shows the computed bellmouth discharge coefficient with and without the fan for approximately the same Mach and Reynolds numbers. It is difficult to match the Mach number exactly since the Mach number is not really constant across the duct upstream of the engine face as shown in the Mach contours in Figure 183. However, as was noted in the earlier bellmouth results, the discharge coefficient is not a strong function of Mach number and therefore for small differences in Mach number the

discharge coefficient is effectively constant. Therefore a comparison of the two cases is valid.

The axial distance in Figure 195 has been normalized by diameter to allow for comparison of the two cases. This figure illustrates the aggregate influence of the presence of the engine fan on bellmouth flow meter performance. Consideration of Figure 195 leads to several conclusions. It is apparent that for the bellmouth with the engine installed the optimum measurement station is shifted upstream considerably to the maximum value location just downstream of the throat. A pressure measurement at the location of maximum discharge coefficient is recommended because it will be least sensitive to errors in tap placement. It is also likely that this region is least sensitive to mass flow variation with different engine speeds.

Because of the large slope in the discharge coefficient versus distance curve, it is advisable not to place the pressure measurement station near the engine face. Furthermore, the magnitude of the pressure fluctuations are much larger close to the engine face, which may make determining the average pressure more difficult. As in the case of a bellmouth with no engine, there is a flat region in the discharge coefficient curve upstream of the throat, but difficulty in accurately determining the cross-sectional flow area in the contraction section and the increased sensitivity to lower Mach number results in a recommendation against placing pressure taps in the bellmouth inlet contraction.

It is difficult to determine an exact accuracy of the calculation given a lack of experimental data to perform a comparison. However an attempt can be made. Integration

of the mass flow at each station resulted in a mass flow variation along the axis of about 0.36 percent. In the bellmouth upstream of the engine face the variation of calculated mass flow is somewhat less at 0.25%. For the 2-D axisymmetric solutions obtained earlier, variations in grid resolution gave results consistently within about 0.07% for the discharge coefficient. Based on these considerations it seems reasonable to expect the present solution method to give results accurate to within 0.4-0.6%. Although the maximum AEDC requirements have been as low as 0.25%, a typical requirement is on the order of 0.5% so these results look very promising.

In summary, a method of analysis has been developed for estimating test facility mass flow metering accuracy through improvement of the WIND code to simulate the presence of a rotating engine face in a bellmouth flow meter. The rotational corrections in the WIND code have been verified to match equivalent non-rotating solutions. The solution has revealed complex pressure behavior in the meter upstream of the engine which requires time averaging to determine the effective discharge coefficient. The discharge coefficient determined in this manner is smooth and well behaved although it varies significantly from a flow meter without an engine present. Finally, it seems promising that this method can be used to determine the proper discharge coefficient and to make recommendations on proper measurement procedures.

5. Summary and Conclusions

The objective of this research was twofold. First, to develop a capability to determine the discharge coefficient of a bellmouth flow meter in compressible flow and to examine the effect of flow conditions and bellmouth geometry on the discharge coefficient.

It was determined that the discharge coefficient can be successfully calculated to an accuracy of 0.5% using the WIND code. This was verified by matching the experimental boundary layer profile data and core flow with numerical predictions based on the WIND code using the SST turbulence model. It was also demonstrated that the Lomax model of turbulence transport gave nearly identical results.

The dependence of discharge coefficient on bellmouth Reynolds number can be quantified. In general the discharge coefficients increased with increasing Reynolds number. This is due to the thinning of the boundary layer with increased Reynolds number. This result is also consistent with historical bellmouth flow meter data.

A dependence of discharge coefficient on Mach number was also demonstrated. In general discharge coefficient increases with increasing Mach number independently of Reynolds number. This increase is caused by changes in streamline curvature in the throat region. At axial locations further downstream from the throat this curvature influence diminishes, and as a result, the dependency of discharge coefficient on Mach number is diminished.

It was also noted that discharge coefficient for compressible flow is lower than for incompressible flow. This is visible in both the experimental data as shown by displacement thickness results and also in the experimental results. There was also previous work that demonstrated this effect.

Discharge coefficients were also shown to be functions of wall static pressure measurement location. This is due primarily to two factors. First, the ideal flow is calculated as a function of wall pressure which changes throughout the bellmouth. Secondly, the displacement thickness changes along the bellmouth wall. These facts impact the static pressure measurement process. To minimize error caused by wall static pressure tap placement, the measurements should be taken where the slope of the axial variation of discharge coefficient is zero or small. If a pressure measurement cannot be obtained at the optimum location due to facility considerations, the capability to calculate the discharge coefficient at any location can be used to make an informed decision on the best measurement process.

It was also noted that the local wall static pressure can be higher than the equivalent 1-D ideal flow static pressure. The discharge coefficient can be greater than one where this condition occurs. The discharge coefficient is seen to represent not an efficiency, but is merely a calibration of apparent flow to actual flow.

Since the discharge coefficient is a function of the pressure measurement process, any errors in the pressure measurement process must be accounted for when comparing to computational results or when using computational results to define the discharge coefficient of a specific flow meter. This was illustrated by the static pressure tap

overpressure correction. Note also that other pressure measurement methods are possible including a freestream pressure probe, all of which may require some form of correction to correctly compare to an idealized computation.

The discharge coefficient with an elliptical inlet profile is a function of the eccentricity of a bellmouth. This dependency is strongest near the throat but drops off rapidly just downstream of the throat. Since the axis ratio in effect represents a rate of contraction for a given diameter ratio, it is reasonable to expect that other bellmouth profiles such as combinations of circular arcs will also demonstrate a dependence on axis ratio.

In comparison however, discharge coefficient was a weak function of diameter ratio for the bellmouth that was examined. This basically amounts to a longer conical inlet, but the elliptical profile is the same and the boundary layer is thinned in the contraction so the flow is very similar.

In the second portion of this research a numerical capability has been developed for computing the flow in a bellmouth flow meter with a rotating blade row installed in the duct outlet, simulating an engine face.

In accomplishing this, the WIND code has been improved to calculate flow in a rotating frame of reference. The modifications consisted of numerous corrections to the flow solver and boundary conditions calculations. Correct formulation of WIND for rotating frames has been demonstrated for both inviscid and viscous flow solutions by comparison to the equivalent inertial frame solutions. The majority of the corrections to

WIND involved proper determination of body forces and extrapolation of rotating components of velocity.

Proper calculation of the flow in a rotating frame for a compressor stage has been demonstrated and compared to experimental data for an experimental NASA compressor stage, Rotor 67. By use of this example, some quantitative conclusions were drawn about the effect of an engine on bellmouth flow meter performance.

The primary upstream effect of the rotating fan is the acceleration of the flow toward the engine face. This results in a drop in average static pressure along the wall approaching the engine and a corresponding drop in the discharge coefficient. In comparison to bellmouth flow without an engine the drop in discharge coefficient is very large, nearly ten percent in close proximity to the engine. Given this large disturbance, it is critical to account for the presence of the engine if proper flow measurement is to be achieved.

Important details were also revealed that impact the determination of the average wall pressure used in calculating the discharge coefficient. As a result of shocks on the leading edge of the blades rotating in the duct, a complex pressure pattern is established upstream of the fan. The complexity arises due to varying angular propagation of pressure disturbances formed at different blade radii. The resulting instantaneous static pressure profile on the wall is very complex. It has several modes of periodicity related to fan rotation speed and the pressure interference pattern just discussed. The amplitude of the pressure waves is itself periodic on the wall along the axis of the duct.

Despite the complexity of the wall static pressure, the time averaged wall static pressure is reasonably smooth and well behaved. This indicates a good possibility of developing a dependable discharge coefficient in the vicinity of the engine inlet. The pressure measurement system must be designed to correctly measure the average pressure at the determined measurement station with the imposed frequency of the disturbance established by engine RPM.

The ideal measurement location for the case considered was about 0.45 throat diameters upstream of the engine face and about 0.15 throat diameters downstream of the throat at the maximum discharge coefficient value. This is about 0.25 diameters farther upstream than the ideal location for the equivalent bellmouth with no fan installed. This indicates that careful consideration must be given to locating the measurement station for a given bellmouth-engine combination.

It seems likely that each engine would have different effect on the duct flow characteristics based on blade profiles and operating conditions. This indicates that the exact performance of each bellmouth-engine combination is unique and should be calculated at least for a finite number of Mach number and Reynolds number combinations. Further comment on this issue is included in the recommendations.

As seen from the earlier calculations of a simple bellmouth meter, the variation with Mach number and Reynolds number is small and fairly linear. This indicates that a table lookup system could be implemented, with the discharge coefficient at a limited number of conditions calculated with the WIND code and entered into the table, and remaining value interpolated from the table look-up.

6. Recommendations for Future Work

The capability to calculate the discharge coefficient in a bellmouth flow meter has been developed and demonstrated. Furthermore, a capability to model the bellmouth flow in the presence of a rotating engine face has been demonstrated. Full application of bellmouth flow meters in a working environment will benefit by additional studies in the following areas.

- Investigation of the boundary layer transition region Reynolds number. The solutions presented in this research assumed a fully turbulent flow. The boundary layer transition region occurs around a Reynolds number of 1×10^6 . Furthermore, this lower Reynolds number is typically associated with lower Mach numbers in the flows of interest, where the discharge coefficient is highly sensitive to static pressure levels. Better determination of discharge coefficient at boundary layer transition requires experimental data at transition Reynolds numbers and corresponding method improvements in calculating transition flows in the WIND code. This could perhaps be facilitated by experimental determination of the transition location, and then specifying the transition in the computation. Intentional tripping of the boundary layer to ensure turbulent flow has also been suggested.

- The influence of other facility installation effects needs to be investigated. In particular, there are a few considerations that have strong potential to affect the boundary layer thickness. The first of these is misalignment of the duct spool pieces. Such misalignments will result in a forward and backward facing step on opposite sides of the duct, which would obviously impact boundary layer thickness. Also present in turbine engine test facilities is the use of a labyrinth seal to isolate thrust forces from the structure. This manifests itself as a small gap or cavity that may also be misaligned. Other possible facility installation issues include converging or diverging duct sections downstream of the bellmouth throat.
- The impact of the pressure tap on the pressure measurement process and variations in pressure tap geometry should be investigated with CFD methods and compared with experiment to determine the effect of the pressure tap on discharge coefficient and to make recommendations on the preferred pressure tap configuration. Also the effects of acoustics and resonance in the pressure lines should be examined to determine if they affect the measured pressure values.
- The impact of flow non-uniformity should be examined if non-uniformities are known to exist. This would include large scale temperature, pressure and velocity distortions as well as turbulence and vorticity levels. Such distortions are known to exist in large scale industrial flow processing.

- It would be advantageous to compare computational results with other well-calibrated compressible bellmouth flow meters. Although there is much previous research on incompressible flow meters, there does not appear to be a body of high quality data for compressible bellmouth flow meters suitable for comparison to computations.
- The WIND code should be carefully examined for further corrections and enhancements in rotating frame calculations. The modifications made in the course of this research were limited to the application described. Modifications have not been implemented for a variety of solution schemes and turbulence models which were not required for this study.
- The rotating frame capability described needs to be extended to include the capability to calculate both a stator row and a rotor row. This would imply some sort of moving frame capability and unsteady calculation. The ability to calculate both a stator and rotor simultaneously would allow for the modeling of engines with inlet guide vanes and the effect of such of bellmouth flow meter performance.
- There is a need to calculate results for a sample engine inlet across a range of Mach and Reynolds numbers and engine speeds. This was not attempted in this study due to the long solution times required. However this would be useful to examine the discharge coefficient profile along the duct to determine optimum measurement locations as well as the need to define bellmouth performance across the expected flow range.

- Finally, it would be useful to compare different engine compressor results. This would give indication as to whether the discharge coefficient is primarily a function of Mach number, Reynolds number, and RPM, or whether it is also sensitive to specific compressor blade geometry and performance.

Bibliography

Bibliography

1. Smith, Robert E., Jr. and Matz, R. J., "A Theoretical Method of Determining Discharge Coefficients for Venturis Operating at Critical Flow Conditions," Journal of Basic Engineering, Transactions of the ASME Series D, Vol. 84, No. 4, Dec. 1962, p. 434 – 446.
2. American Society of Mechanical Engineers Research Committee on Fluid Meters, Fluid Meters, Their Theory and Application, 5th ed, New York: The American Society of Mechanical Engineers, 1959.
3. American Society of Mechanical Engineers Research Committee on Fluid Meters, Application, Part II of Fluid Meters, 6th ed, New York: The American Society of Mechanical Engineers, 1972.
4. Shapiro, A.H. and Smith, R. D., "Friction Coefficients in the Inlet Length of Smooth Round Tubes," NACA Technical Note No. 1785, November 1948.
5. Langhaar, Henry L., "Steady Flow in the Transition Length of a Straight Tube," Journal of Applied Mechanics, vol. 9, no. 2, 1942, p. A-55.
6. Rivas, M. A. Jr., and Shapiro, A. H., "On the Theory of Discharge Coefficients for Rounded-Entrance Flowmeters and Venturis", Transactions of the ASME, April, 1956, pp. 489 – 497.
7. Cotton, K. C., and Westcott, J. C., "Throat Tap Nozzles Used for Accurate Flow Measurements," ASME Journal of Engineering for Power, Oct. 1960, pp. 247 – 263.
8. Rayle, R. E., "Influence of Orifice Geometry on Static Pressure Measurements," ASME Paper 59-A-234, December 1959.
9. Hall, G.W., "Analytical Determination of the Discharge Characteristics of Cylindrical-Tube Orifices," Journal of Mechanical Engineering Science, Vol. 5, No. 1, 1963, pp. 91 – 97.
10. Leutheusser, Hans. J., "Flow Nozzles With Zero Beta Ratio," Transactions of the ASME, Journal of Basic Engineering, September 1964, pp. 538 - 542.
11. Soundrananyagam, S., "An Investigation Into the Performance of Two ISA Metering Nozzles of Finite and Zero Area Ratio," Transactions of the ASME, Journal of Basic Engineering, June 1965, p. 525 - 532.

12. Benedict, Robert P., "Most Probable Discharge Coefficient for ASME Flow Nozzles," Transactions of the ASME, Journal of Basic Engineering, December 1966, pp. 734 - 744.
13. Cotton, K. C., Carcich, J. A., and Schofield, P., "Experience with Throat-Tap Nozzles for Accurate Flow Measurement," Transactions of the ASME, Journal of Engineering for Power, April 1972, pp. 133 - 141.
14. Miller, R. W., and Kneisel, O., "A Comparison Between Orifice and Flow Nozzle Laboratory Data and Published Coefficients," Transactions of the ASME, Journal of Fluids Engineering, June 1974, pp. 139 - 149.
15. Benedict, R. P., and Wyler, J. S., "A Generalized Discharge Coefficient for Differential Pressure Type Fluid Meters," Transactions of the ASME, Journal of Engineering for Power, October 1974, pp. 440 - 447.
16. Benedict, R. P., and Wyler, J. S., "Comparisons Between Throat and Pipe Wall Tap Nozzles," Transactions of the ASME, Journal of Engineering for Power, October 1975, pp. 569 - 575.
17. Benedict, R. P., and Wyler, J. S., "Analytical and Experimental Studies of ASME Flow Nozzles," ASME Journal of Fluids Engineering, September 1978, pp. 265 - 275.
18. Benedict, R.P., "Generalized Fluid Meter Discharge Coefficient Based Solely on Boundary Layer Parameters," Transactions of the ASME, Journal of Engineering for Power, October 1979, pp. 572 - 575.
19. Benedict, R. P., "The Plenum Inlet Discharge Coefficient of an ASME nozzle," Flow, Its measurement and control in science and industry; Proceedings of the Second Symposium, Instrument Society of America, 1981, pp. 363-372.
20. Murdock, J. W., and Keyser, D. R., "Theoretical Basis for Extrapolation of Calibration Data of PTC 6 Throat Tap Nozzles," Transactions of the ASME, Journal of Engineering for Gas Turbines and Power, April 1991, Vol. 113, pp. 228 - 232.
21. Murdock, J. W., and Keyser, D. R., "A Method for the Extrapolation of Calibration Data of PTC 6 Throat Tap Nozzles," Transactions of the ASME, Journal of Engineering for Gas Turbines and Power, April 1991, Vol. 113, p. 233 -241.
22. Smith, Stephen C., "Airflow Calibration of a Bellmouth Inlet for Measurement of Compressor Airflow in Turbine-Powered Propulsion Simulators," NASA Technical Memorandum 84399, November 1985.

23. Reimer, Robert M., "Determination of ASME Nozzle Flow Coefficients by Thrust Measurement," Transactions of the ASME, Journal of Basic Engineering, December 1965, pp. 1058 - 1062.
24. Smith, Jr., R. E., and Matz, R. J., "Performance Characteristic of an 8-In-Dia ASME Nozzle Operating at Compressible and Incompressible Conditions," Transactions of the ASME, Journal of Fluids Engineering, December 1973, pp. 542 - 550.
25. Wehofer, S., and Moger, W. C., "Analysis and Computer Program for Evaluation of Airbreathing Propulsion Exhaust Nozzle Performance," AEDC-TR-73-29, May 1973.
26. Lahti, D. J., Theory and Experiments on Subcritical Compressible Gas Flow Metering, Diss., Univ. of Cincinnati, Cincinnati, OH, Aug. 1990.
27. Lahti, D. J., and Hamed, A., "Verification of the Theoretical Discharge Coefficient of a Subcritical Airflow Meter," Journal of Propulsion and Power, Vol. 9, No. 4, July-Aug. 1993, pp. 615 - 621.
28. Reed, J. A., Hiers, R. S., Jr., Turrentine, W. A., Jr., "Improvement of Flow Quality in Turbine Engine Test Cells by the Elimination of Bellmouth-induced Core-flow Vortices," AIAA Paper 95-2391, AIAA, ASME, SAE, and ASEE, Joint Propulsion Conference and Exhibit, 31st, San Diego, CA, July 10-12, 1995.
29. Beale, D. K., Hand, T. L., and Seaborn, C. L., "Development of a Bellmouth Airflow Measurement Technique for Turbine Engine Ground Test Facilities," AIAA 2001-3676, Presented at the AIAA/ASME/SAE/ASEE 37th Joint Propulsion Conference and Exhibit, Salt Lake City, UT, July 8-11, 2001.
30. Baldwin, B. S., and Lomax, H., "Thin Layer Approximations and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, 1978.
31. Baldwin, B. S., and Barth, T. J., "A One-Equation Turbulence Transport Model for High Reynolds Number Wall-Bounded Flows," NASA TM 192847, 1990.
32. Menter, F. R., "Zonal Two Equation $\kappa - \omega$ Turbulence Models for Aerodynamic Flows," AIAA Paper 93-2906, 1993.
33. The NPARC Alliance, NASA Glenn Research Center, USAF Arnold Engineering Development Center, WIND User's Guide, April 2001.
34. Hirsch, C., Numerical Computation of Internal and External Flows, Volume 2, Computational Methods for Inviscid and Viscous Flows, Chichester, John Wiley and Sons, 1990.

35. Rayle, R. E., "Influence of Orifice Geometry on Static Pressure Measurements," ASME Paper 59-A-234, December, 1959.
36. Franklin, R. E. and Wallace, J. M., "Absolute Measurement of Static-Hole Error Using Flush Transducers," Journal of Fluid Mechanics, Vol. 42, Part 1, 1970, pp. 33 - 48.
37. Ucer, A. S., Stow, P., and Hirsch, Ch., Thermodynamics and Fluid Mechanics of Turbomachinery, Vol. 1, Boston, Martinus Nijhoff Publishers, 1995.
38. Strazisar, A. J., Wood, J.R., Hathaway, M.D., and Suder, K.L., "Laser Anemometer Measurements in a Transonic Axial-Flow Fan Rotor," NASA Technical Paper 2879, November 1989.

Appendices

Appendix A. Tables

Table 1.
Mach and Reynolds Number Conditions for the WIND-based CFD Simulations of
Bellmouth Flows

Mach No	Ptot	Ps	Tt	Re
0.2	16.24	15.80	530	2.00e6
0.2	24.36	23.69	530	3.00e6
0.2	31.67	30.80	530	3.90e6
0.2	46.70	45.41	530	5.75e6
0.2	81.21	78.98	530	10.00e6
0.2	162.42	157.96	530	20.00e6
0.31	10.74	10.05	530	2.00e6
0.31	16.11	15.07	530	3.00e6
0.31	20.94	19.59	530	3.90e6
0.31	30.88	28.89	530	5.75e6
0.31	53.70	50.24	530	10.00e6
0.31	107.40	100.48	530	20.00e6
0.4	8.56	7.66	530	2.00e6
0.4	12.84	11.50	530	3.00e6
0.4	16.69	14.95	530	3.90e6
0.4	24.60	22.03	530	5.75e6
0.4	42.79	38.32	530	10.00e6
0.4	85.57	76.64	530	20.00e6
0.4	171.15	153.28	530	40.00e6
0.5	7.11	6.00	530	2.00e6
0.5	10.67	9.00	530	3.00e6
0.5	13.87	11.69	530	3.90e6
0.5	20.45	17.24	530	5.75e6
0.5	35.57	29.98	530	10.00e6
0.5	71.14	59.97	530	20.00e6
0.5	142.27	119.94	530	40.00e6
0.6	6.21	4.87	530	2.00e6
0.6	9.31	7.30	530	3.00e6
0.6	12.10	9.49	530	3.90e6
0.6	17.84	13.99	530	5.75e6
0.6	31.03	24.33	530	10.00e6
0.6	62.07	48.66	530	20.00e6
0.6	124.13	97.32	530	40.00e6
0.7	5.61	4.04	530	2.00e6
0.7	8.41	6.07	530	3.00e6
0.7	10.94	7.89	530	3.90e6
0.7	16.12	11.63	530	5.75e6
0.7	28.05	20.22	530	10.00e6
0.7	56.10	40.44	530	20.00e6
0.7	112.19	80.88	530	40.00e6

Appendix B. Figures

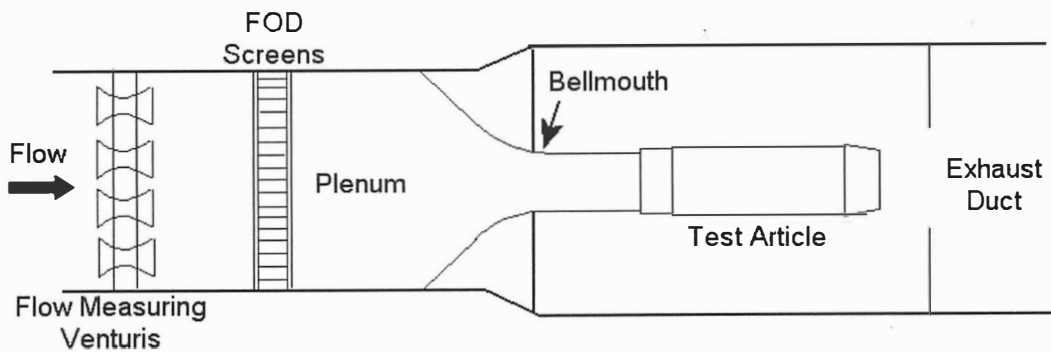


Figure 1. Typical Turbine Engine Test Cell Showing Location of Flow Measuring Venturis.

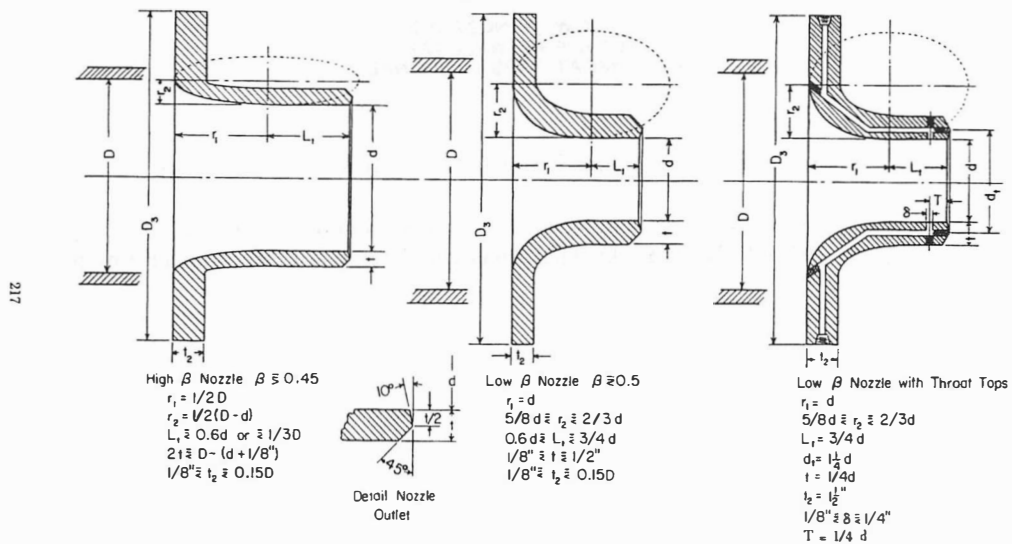


Figure 2. ASME Flow Meter Recommended Geometry.

Source: American Society of Mechanical Engineers Research Committee on Fluid Meters, Application, Part II of Fluid Meters, 6th ed, New York: The American Society of Mechanical Engineers, 1972.

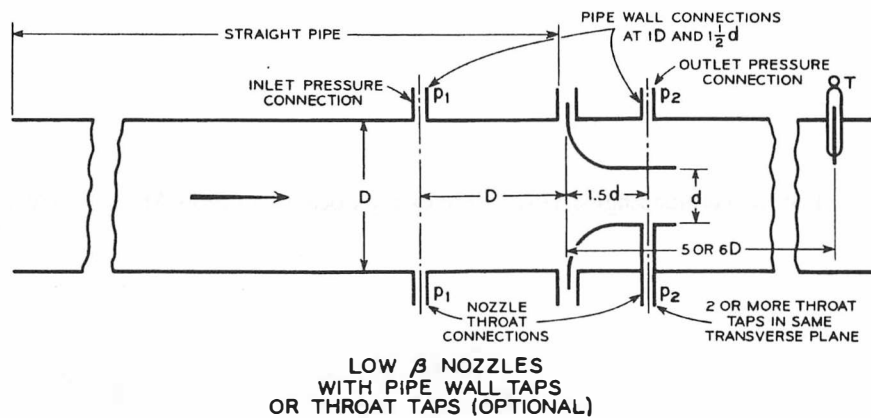


Figure 3. Location of Pressure Taps Used With ASME Flow Nozzles.

Source: American Society of Mechanical Engineers Research Committee on Fluid Meters, Application, Part II of Fluid Meters, 6th ed, New York: The American Society of Mechanical Engineers, 1972.

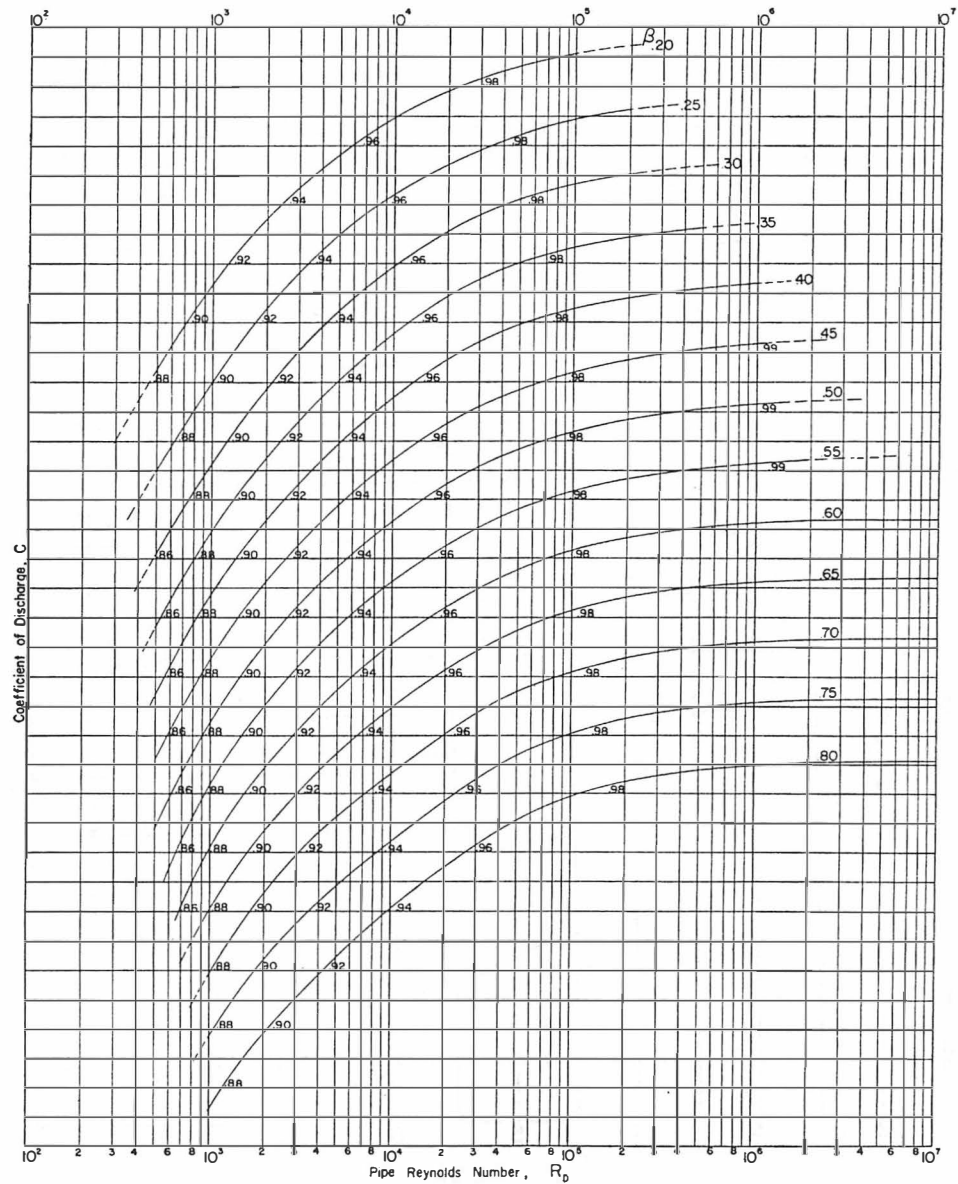


Figure 4: ASME Flow Nozzle Coefficients.

Source: American Society of Mechanical Engineers Research Committee on Fluid Meters, Fluid Meters, Their Theory and Application, 5th ed, New York: The American Society of Mechanical Engineers, 1959.

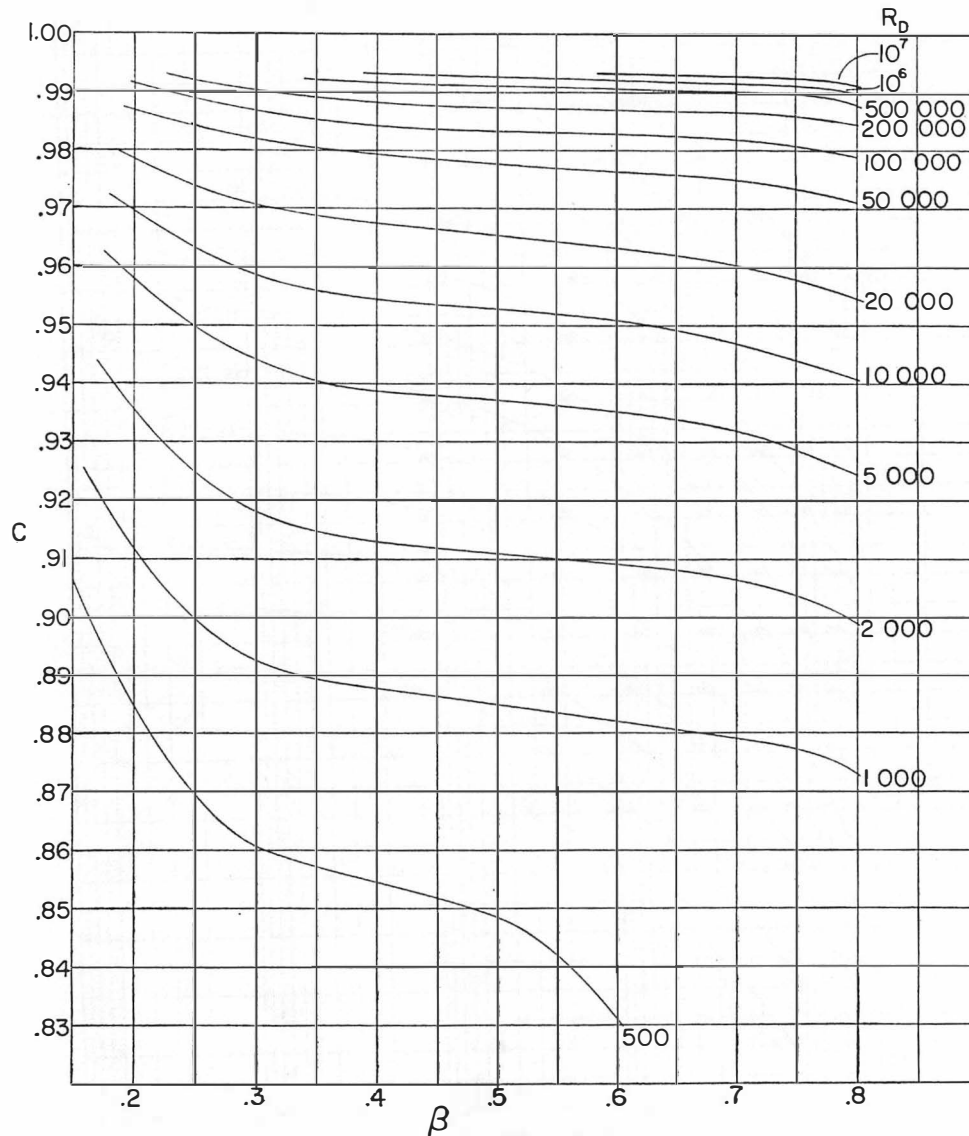


Figure 5. ASME Flow Nozzle Discharge Coefficient as a Function of Beta Ratio.

Source: American Society of Mechanical Engineers Research Committee on Fluid Meters, Fluid Meters, Their Theory and Application, 5th ed, New York: The American Society of Mechanical Engineers, 1959.

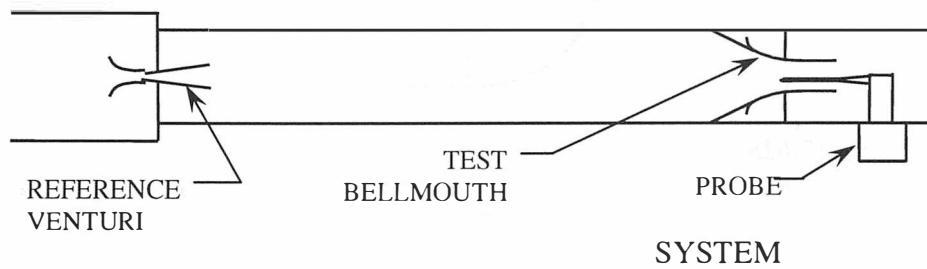


Figure 6. Bellmouth Experiment Schematic..

Source: Beale, D. K., Hand, T. L., and Sebourn, C. L., "Development of a Bellmouth Airflow Measurement Technique for Turbine Engine Ground Test Facilities," AIAA 2001-3676, Presented at the AIAA/ASME/SAE/ASEE 37th Joint Propulsion Conference and Exhibit, Salt Lake City, UT, July 8-11, 2001.

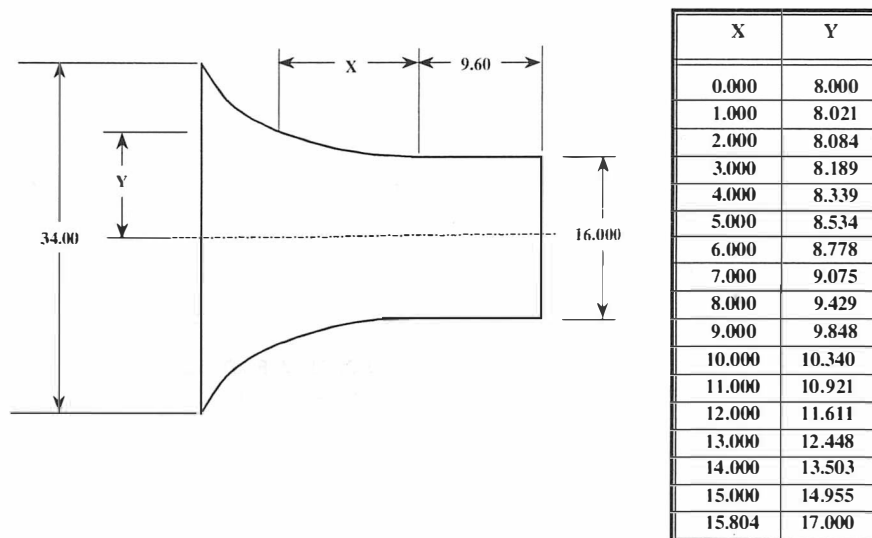


Figure 7. Bellmouth Contraction Geometry.

Source: Beale, D. K., Hand, T. L., and Sebourn, C. L., "Development of a Bellmouth Airflow Measurement Technique for Turbine Engine Ground Test Facilities," AIAA 2001-3676, Presented at the AIAA/ASME/SAE/ASEE 37th Joint Propulsion Conference and Exhibit, Salt Lake City, UT, July 8-11, 2001.

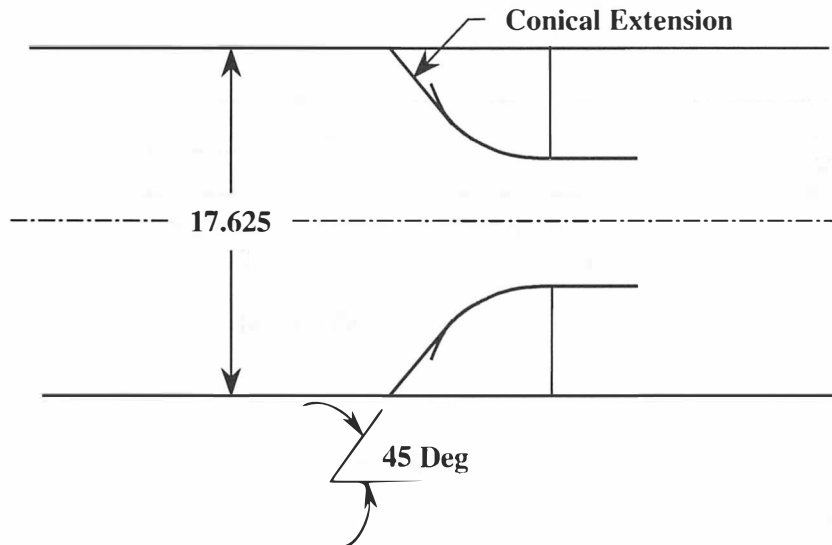


Figure 8. Installed Experimental Bellmouth Configuration.

Source: Beale, D. K., Hand, T. L., and Sebourn, C. L., "Development of a Bellmouth Airflow Measurement Technique for Turbine Engine Ground Test Facilities," AIAA 2001-3676, Presented at the AIAA/ASME/SAE/ASEE 37th Joint Propulsion Conference and Exhibit, Salt Lake City, UT, July 8-11, 2001.

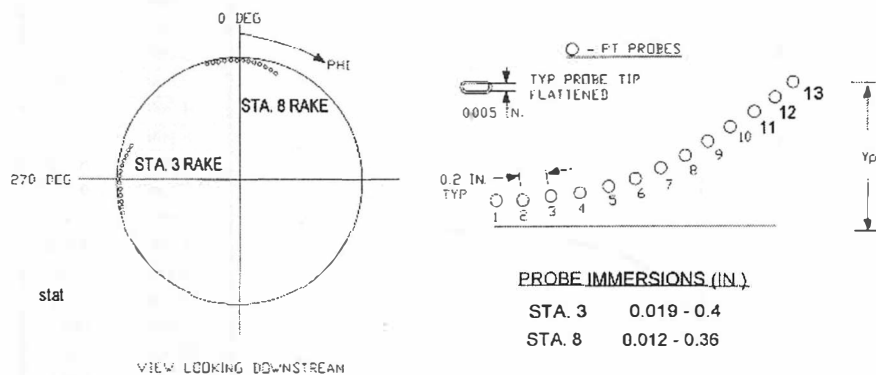


Figure 9. Boundary Layer Rake Detail.

Source: Beale, D. K., Hand, T. L., and Sebourn, C. L., "Development of a Bellmouth Airflow Measurement Technique for Turbine Engine Ground Test Facilities," AIAA 2001-3676, Presented at the AIAA/ASME/SAE/ASEE 37th Joint Propulsion Conference and Exhibit, Salt Lake City, UT, July 8-11, 2001.

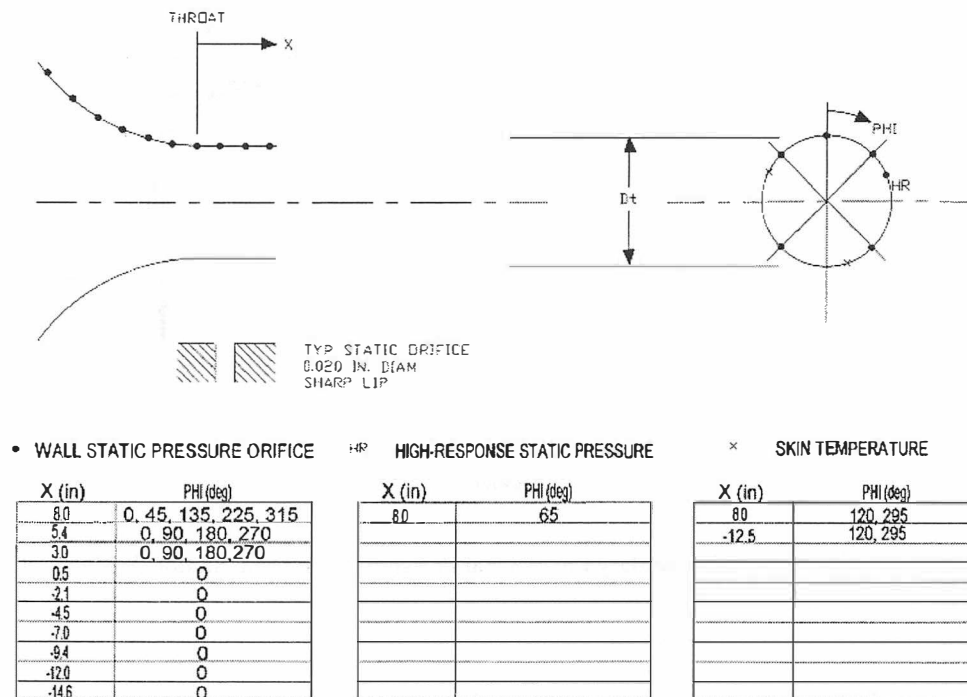


Figure 10. Surface Measurement Detail.

Source: Beale, D. K., Hand, T. L., and Sebourn, C. L., "Development of a Bellmouth Airflow Measurement Technique for Turbine Engine Ground Test Facilities," AIAA 2001-3676, Presented at the AIAA/ASME/SAE/ASEE 37th Joint Propulsion Conference and Exhibit, Salt Lake City, UT, July 8-11, 2001.

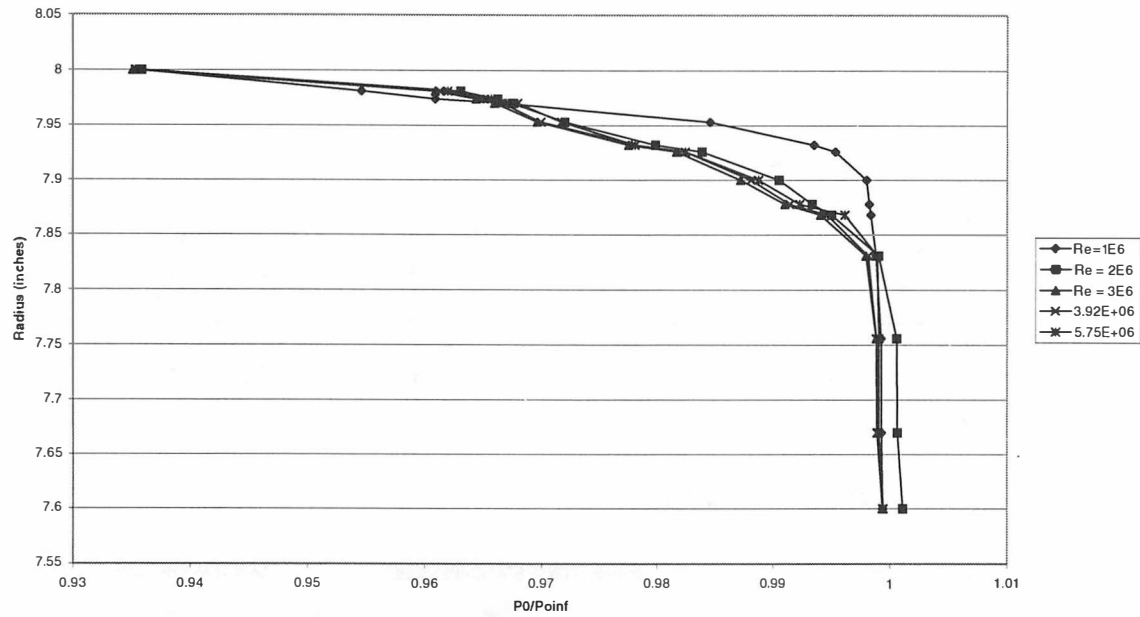


Figure 11. Experimental Boundary Layer Total Pressure Probe Data, Station 3, Mach = 0.31.

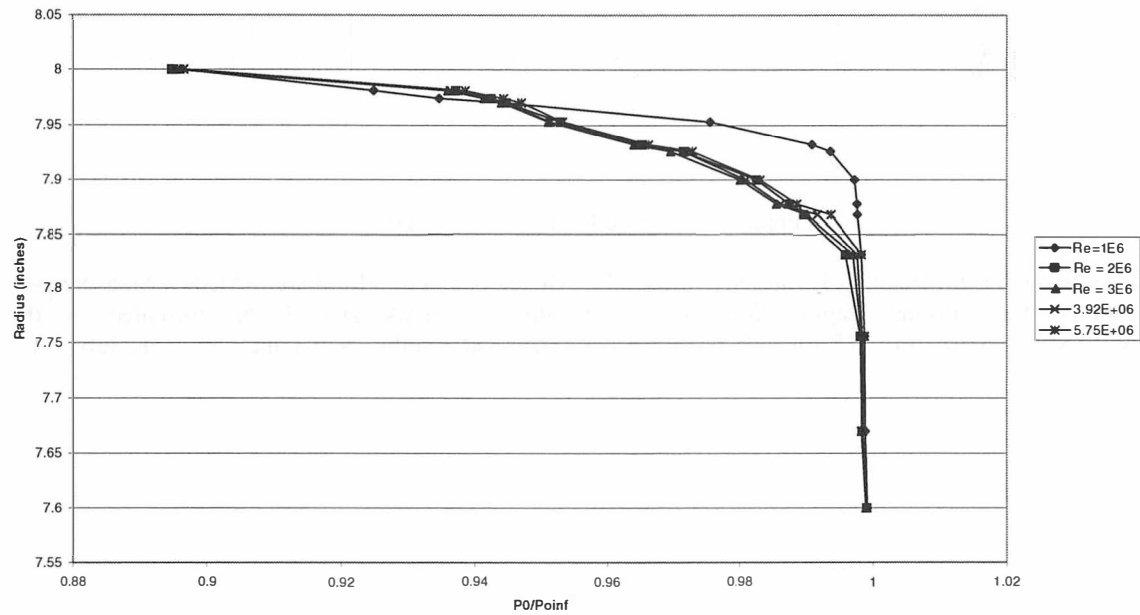


Figure 12. Experimental Boundary Layer Total Pressure Probe Data, Station 3, Mach = 0.4.

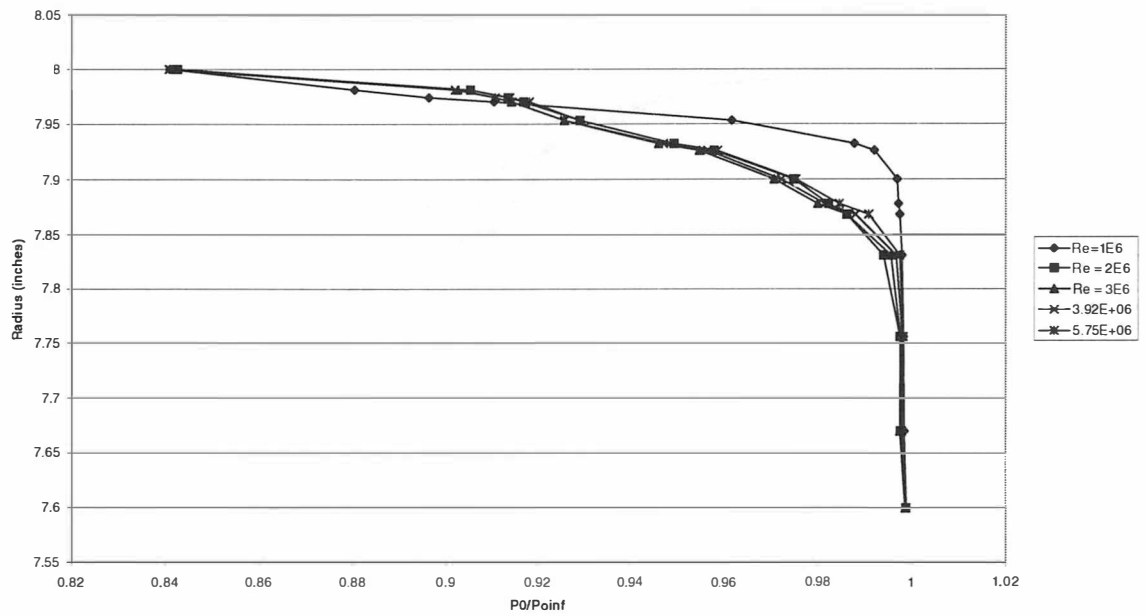


Figure 13. Experimental Boundary Layer Total Pressure Probe Data, Station 3, Mach = 0.5.

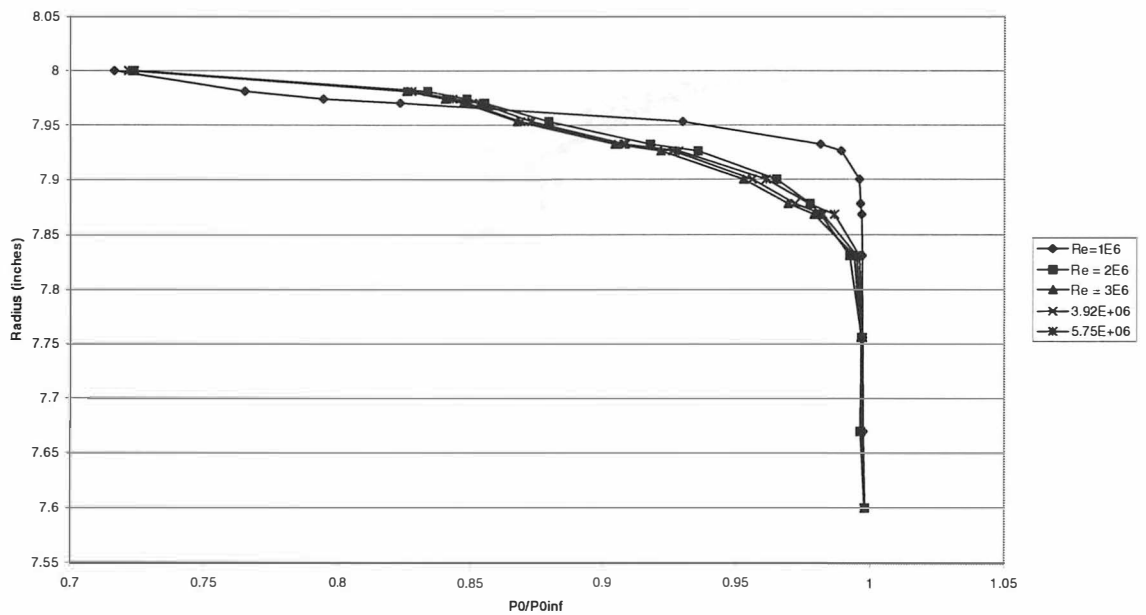


Figure 14. Experimental Boundary Layer Total Pressure Probe Data, Station 3, Mach = 0.7.

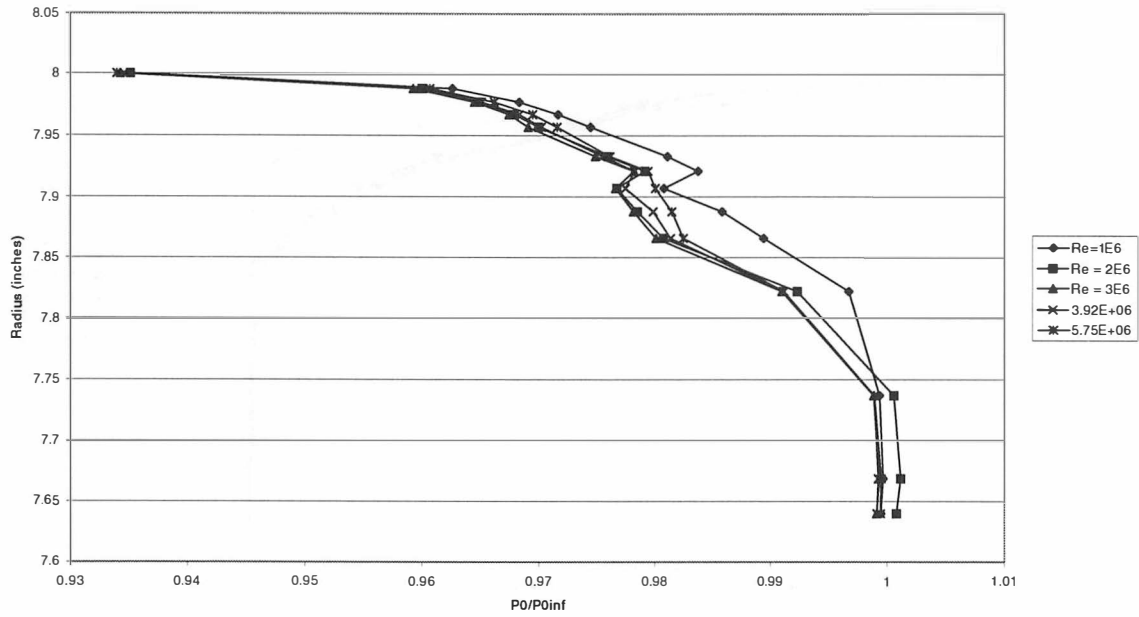


Figure 15. Experimental Boundary Layer Total Pressure Probe Data, Station 8, Mach = 0.31.

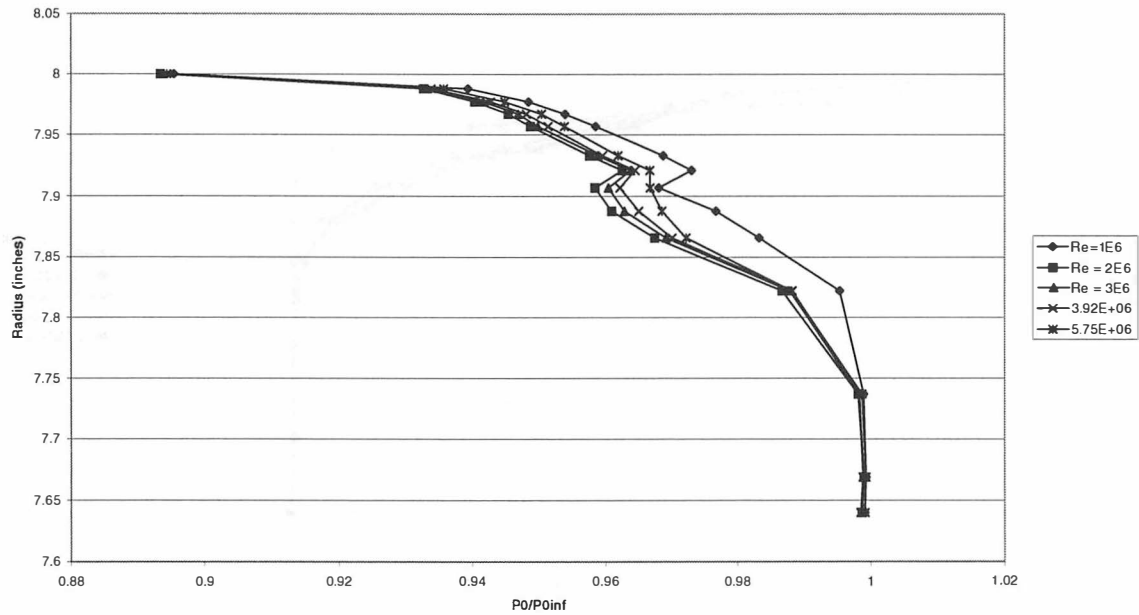


Figure 16. Experimental Boundary Layer Total Pressure Probe Data, Station 8, Mach = 0.4.

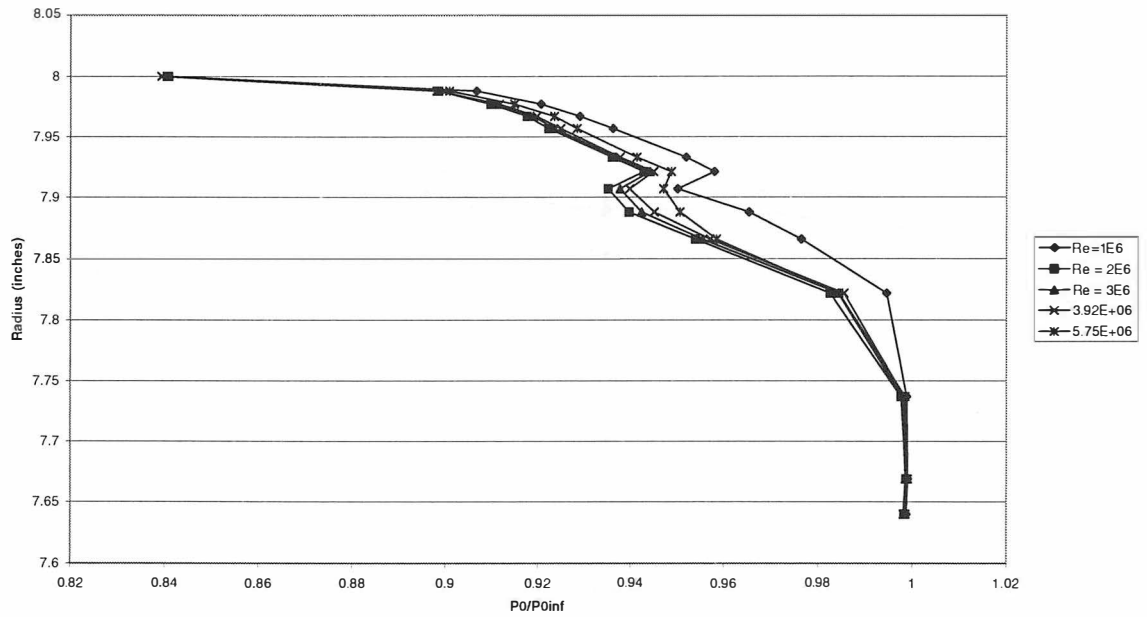


Figure 17. Experimental Boundary Layer Total Pressure Probe Data, Station 8, Mach = 0.5.

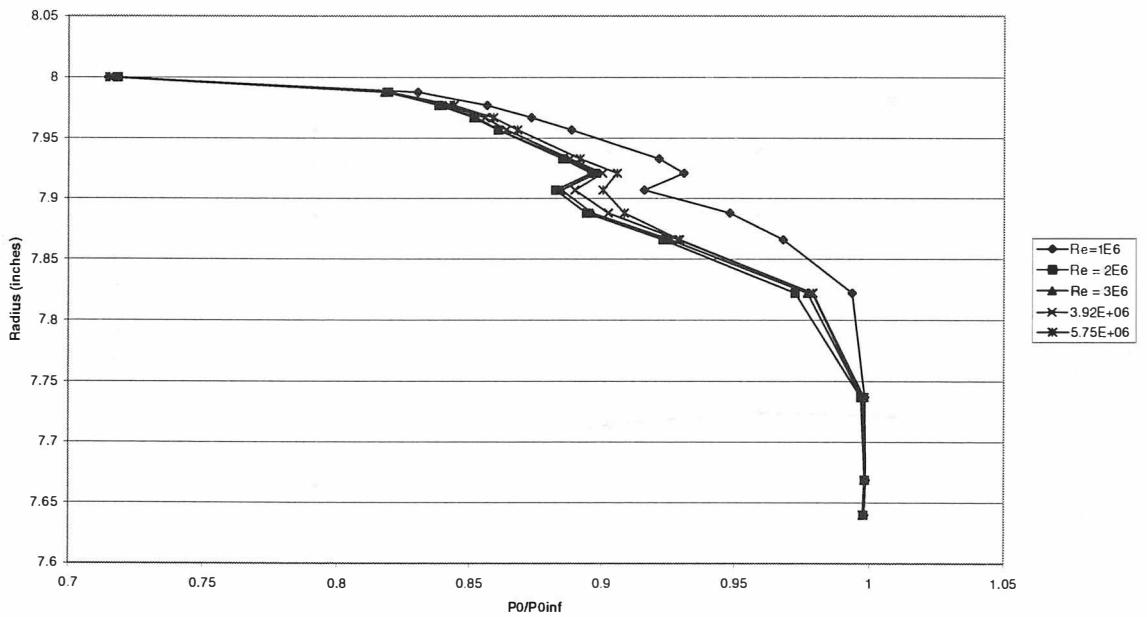


Figure 18. Experimental Boundary Layer Total Pressure Probe Data, Station 8, Mach = 0.7.

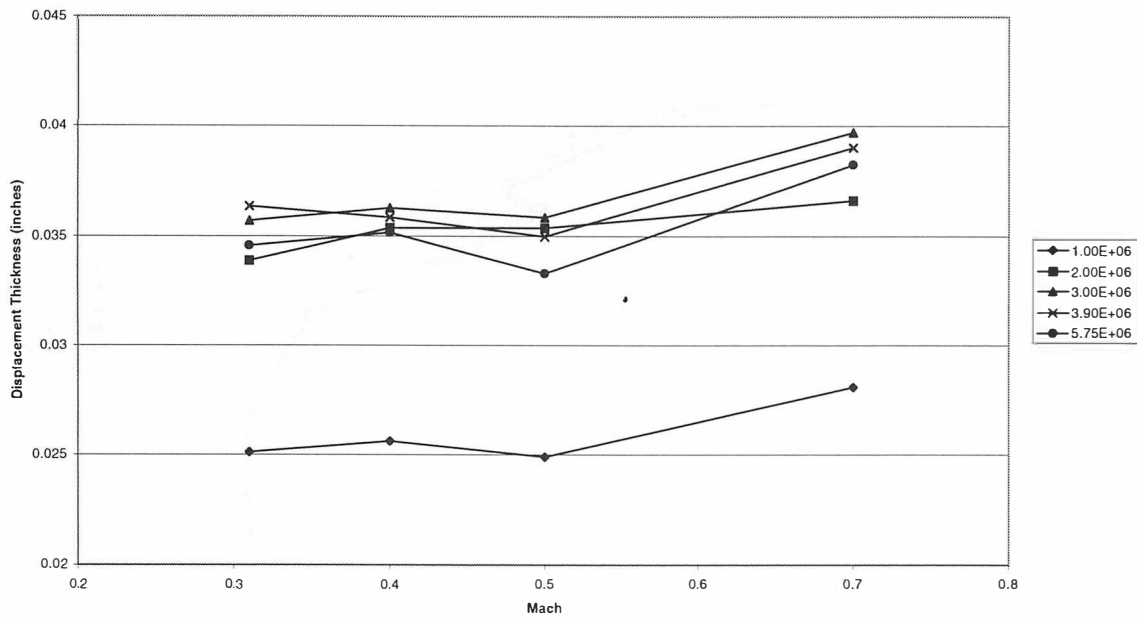


Figure 19. Experimental Displacement Thickness at Station 3 for Various Reynolds No.

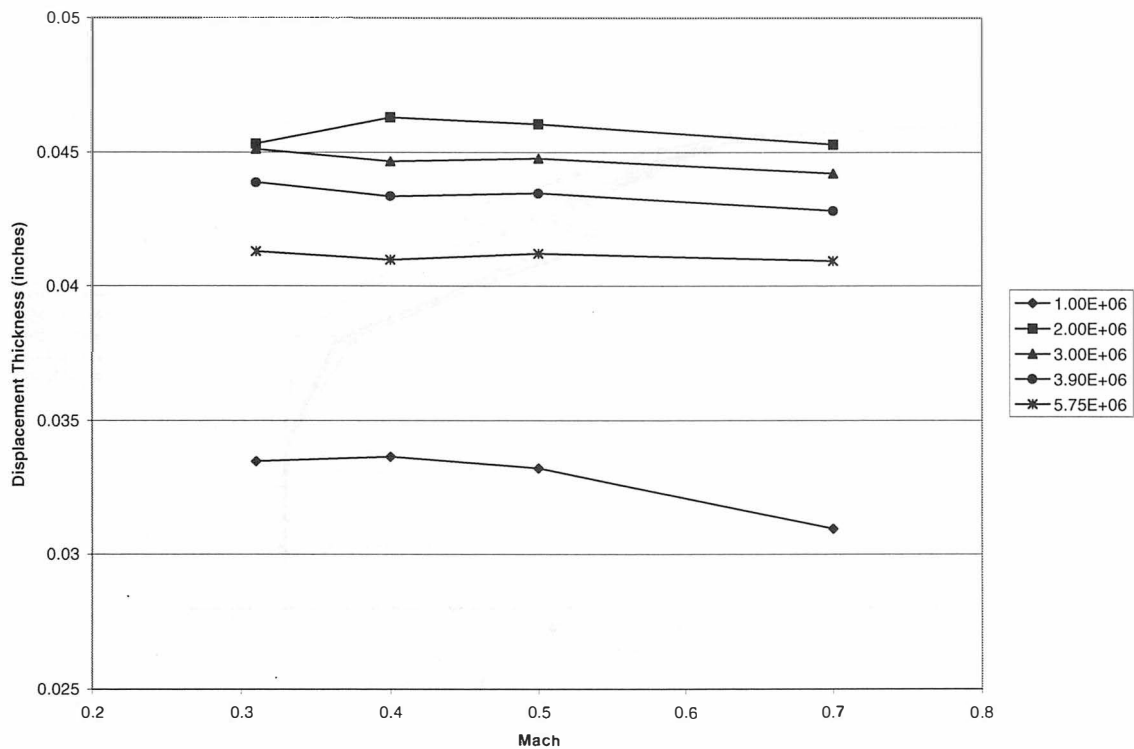


Figure 20. Experimental Displacement Thickness at Station 8 for Various Reynolds No.

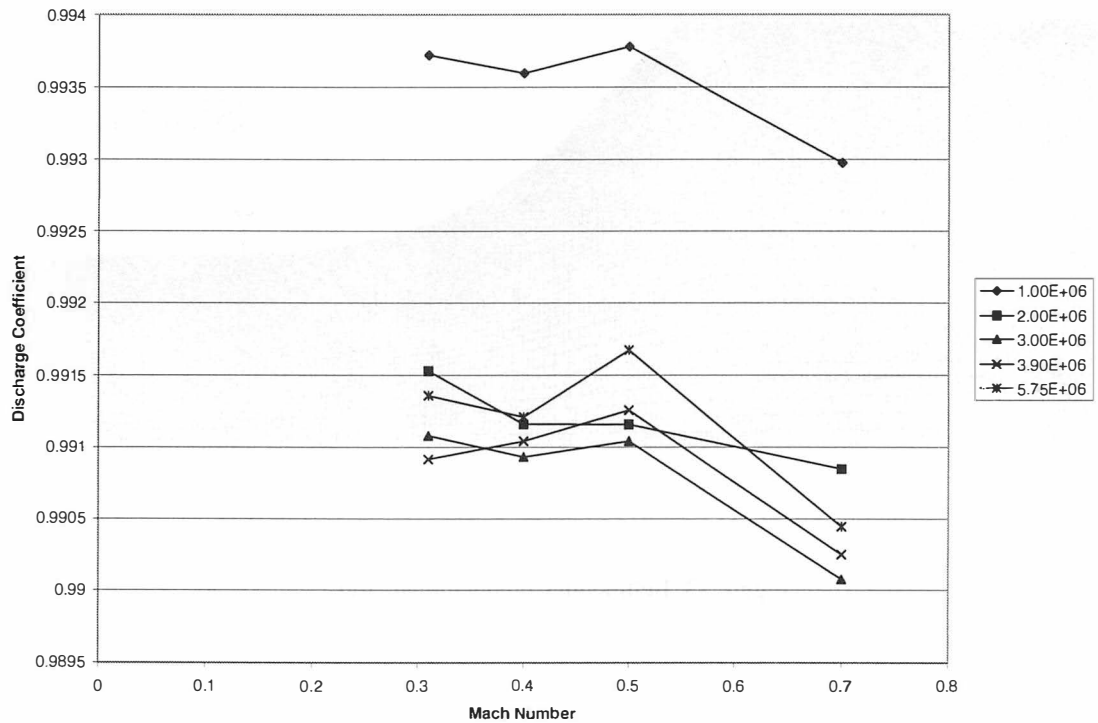


Figure 21. Experimental Discharge Coefficient at Station 3 Based on Displacement Thickness.

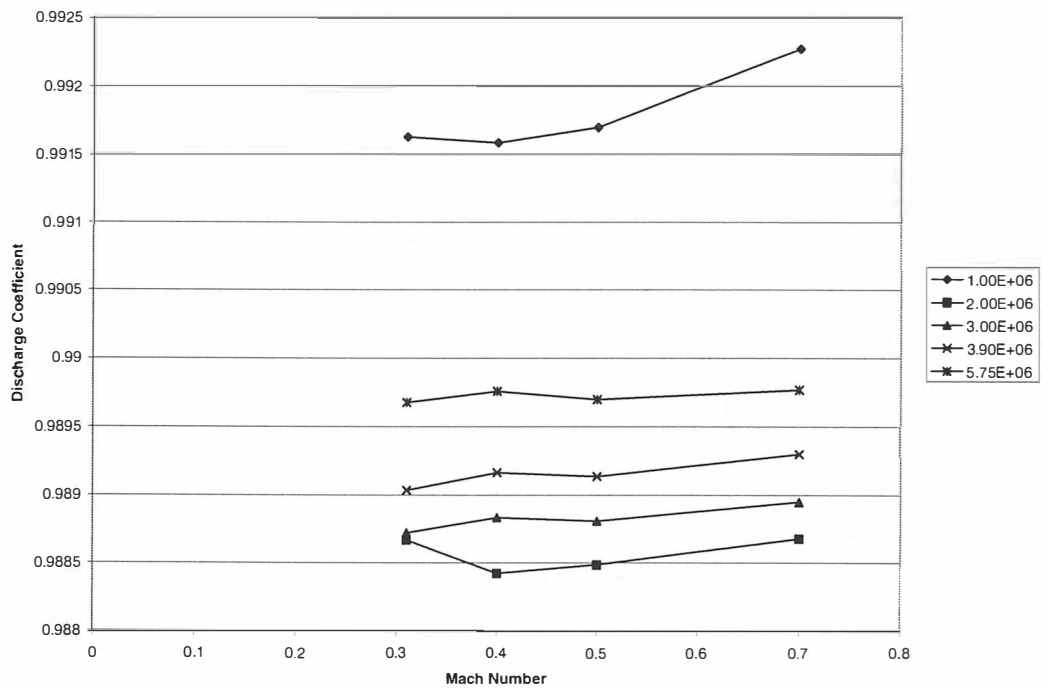


Figure 22. Experimental Discharge Coefficient at Station 8 Based on Displacement Thickness.

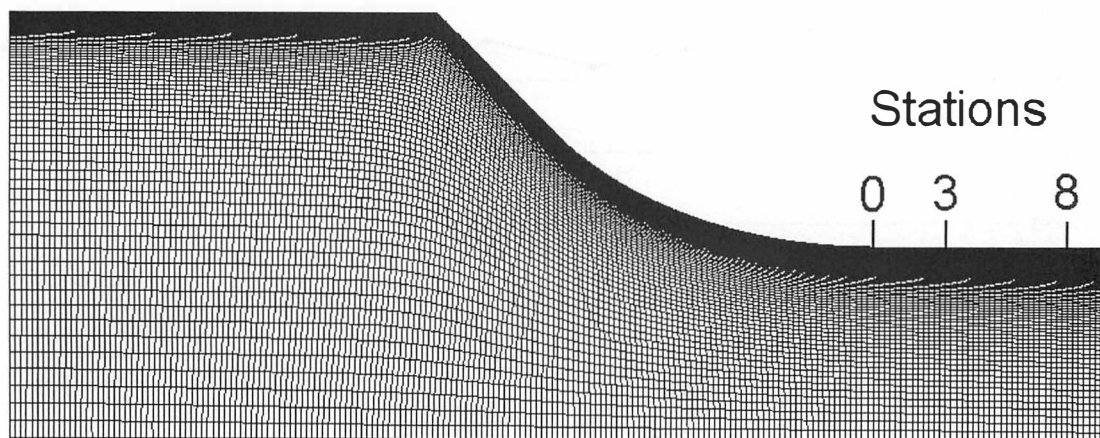


Figure 23. Bellmouth Computational Grid.

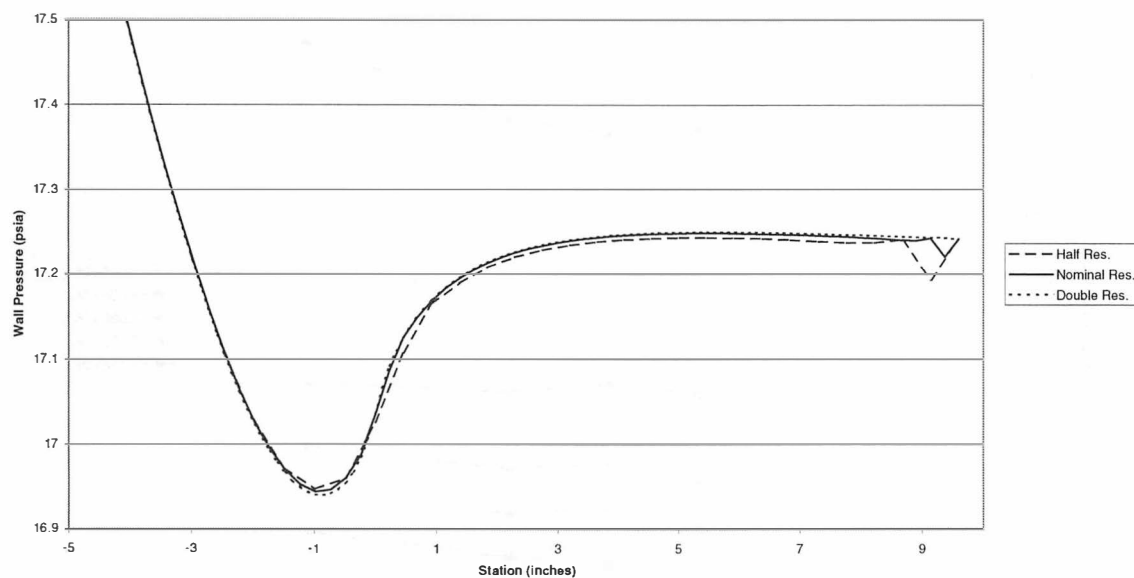


Figure 24. Computed Wall Static Pressure at Mach = 0.5 and Reynolds No. = 5.75E6 for Various Grid Resolutions.

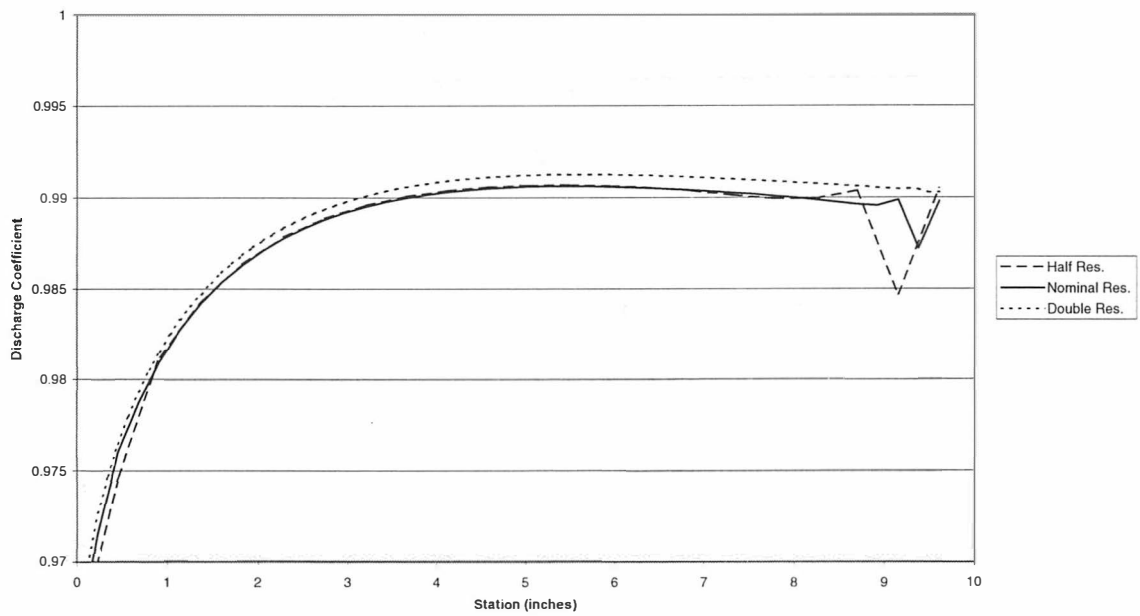


Figure 25. Computed Discharge Coefficient in the Throat Region, Mach = 0.5, Reynolds No. = 5.65E6 for Various Grid Resolutions.

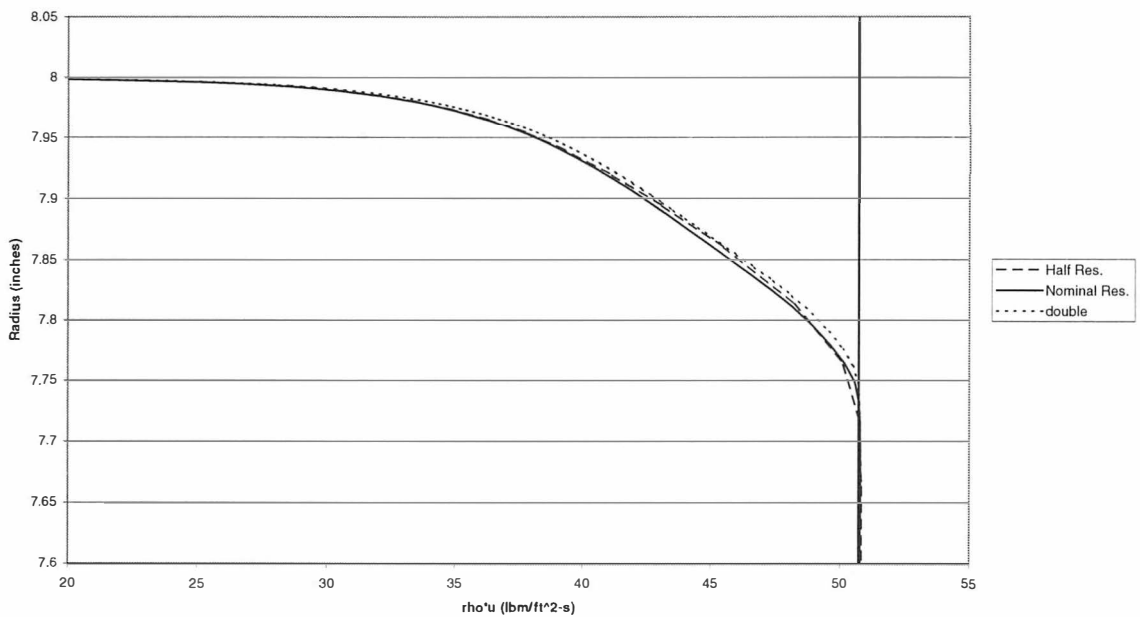


Figure 26. Computed Boundary Layer Mass Flux at Station 8, M=0.5, Reynolds No. = 5.75E6 for Various Grid Resolutions.

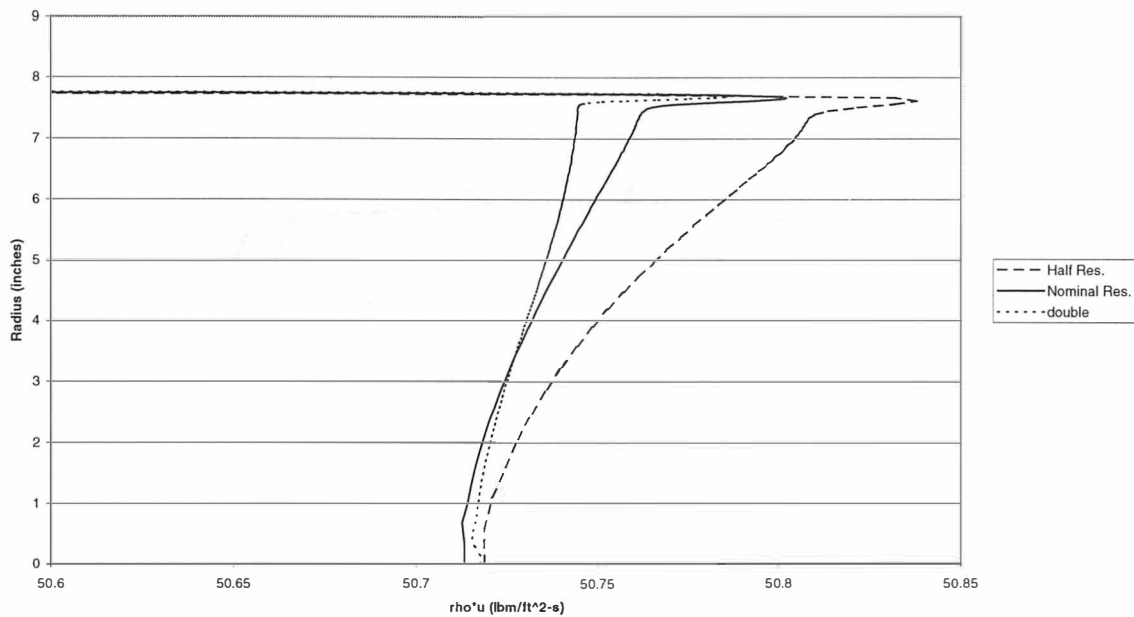


Figure 27. Computed Mass Flux at Station 8, Mach =0.5, Reynolds No. = 5.75E6 for Various Grid Resolutions.

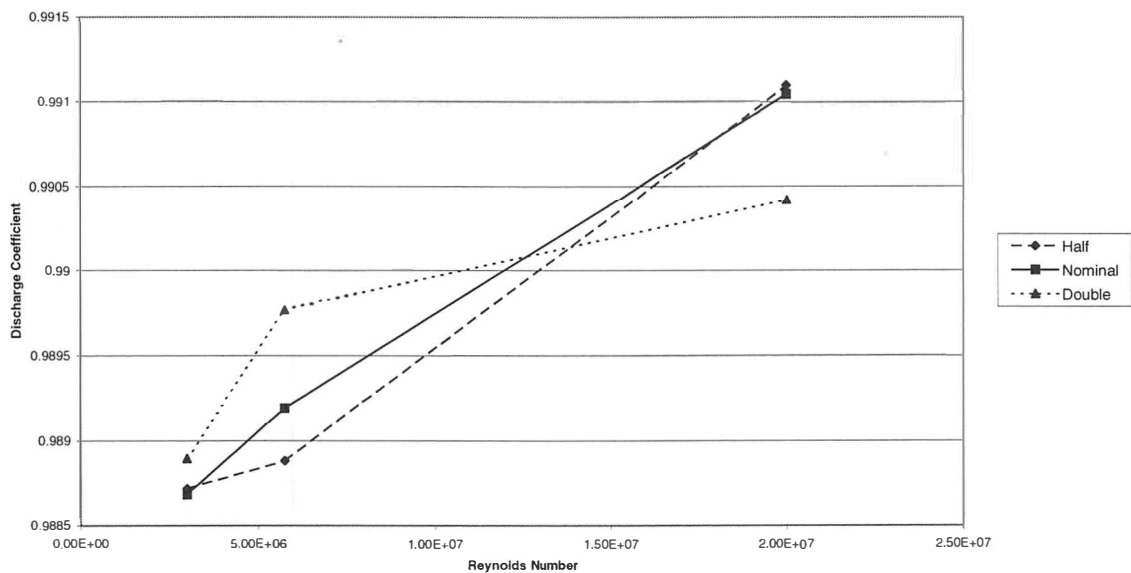


Figure 28. Discharge Coefficient at Station 3 Computed Using Various Grid Resolutions, Mach = 0.5.

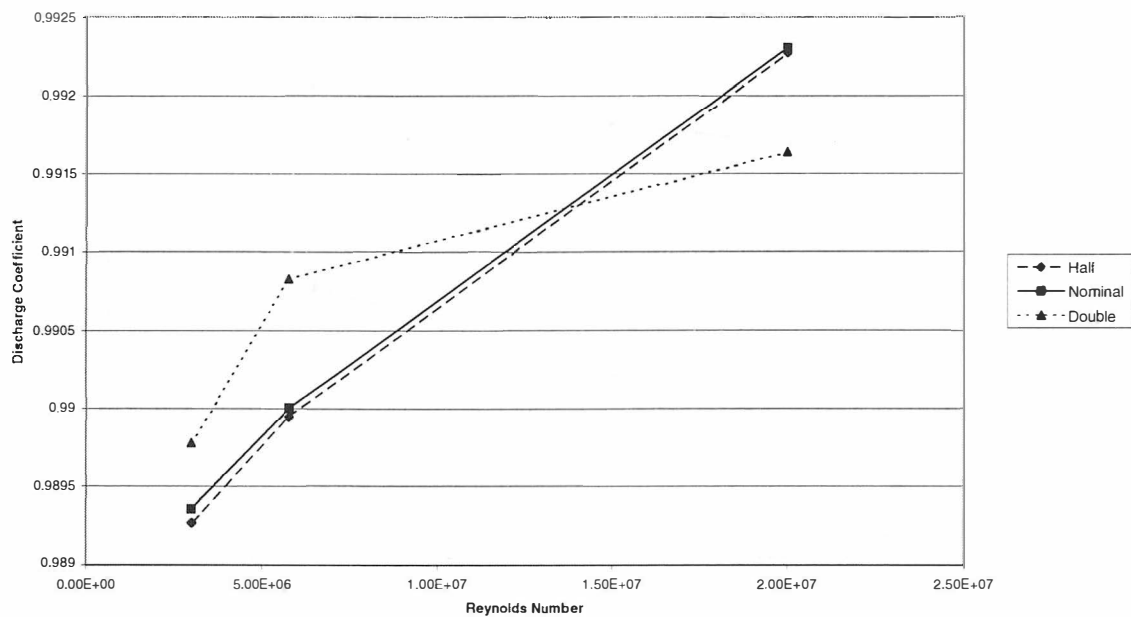


Figure 29. Discharge Coefficient at Station 8 Computed Using Various Grid Resolutions, Mach = 0.5.

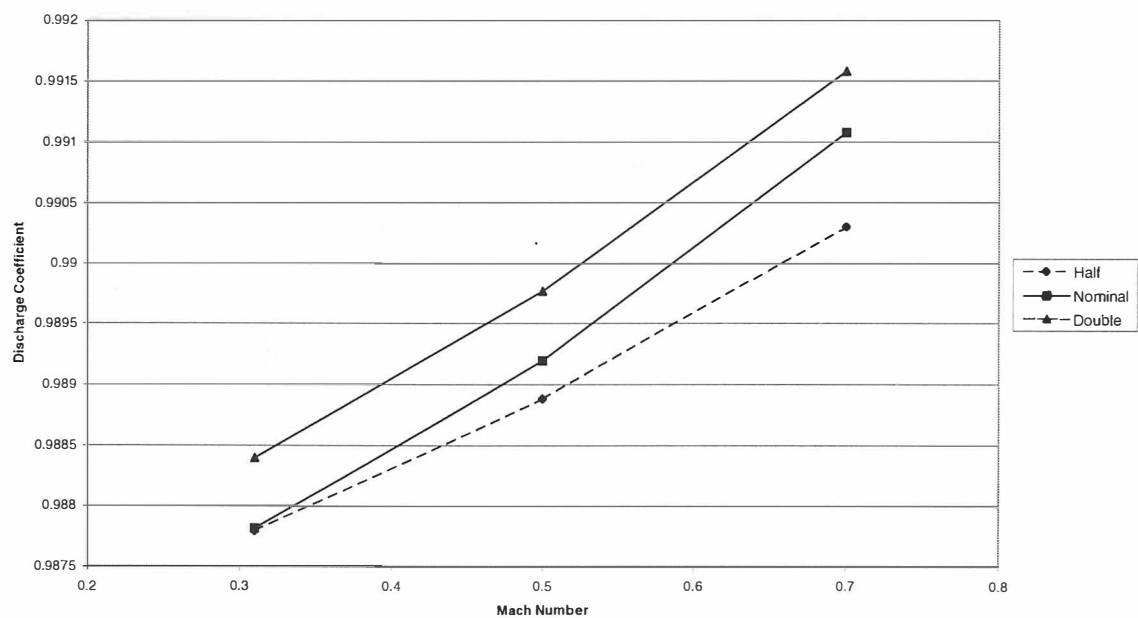


Figure 30. Discharge Coefficient at Station 3 Computed Using Various Grid Resolutions, Re = 5.75E6.

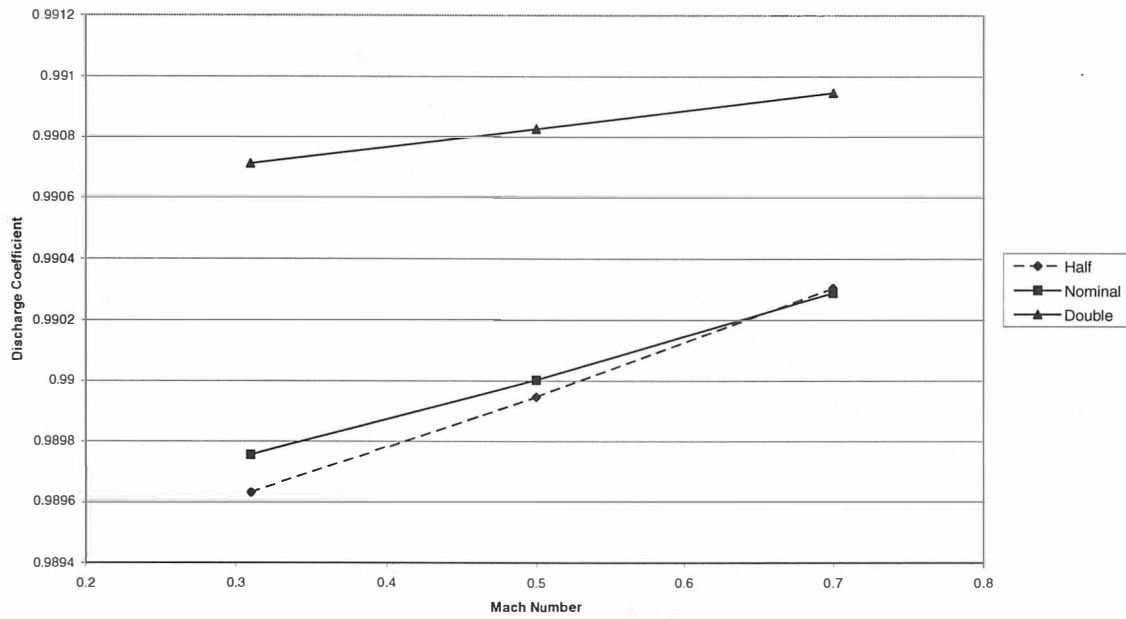


Figure 31. Discharge Coefficient at Station 8 Computed Using Various Grid Resolutions, $Re = 5.75E6$.

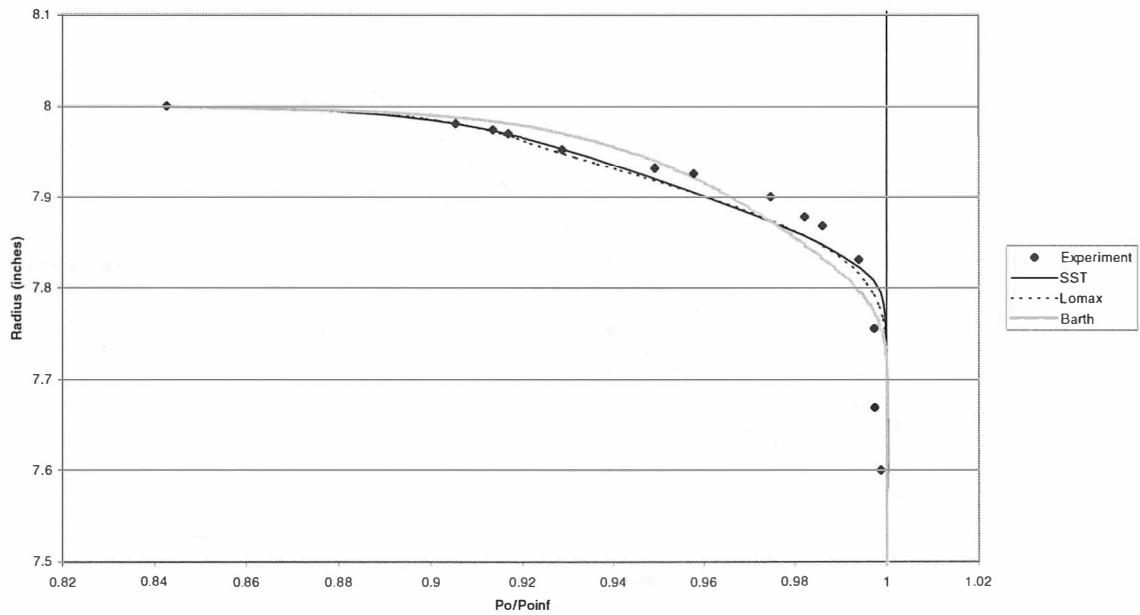


Figure 32. Comparison of Turbulence Model Results for Total Pressure Profile to Experimental Data at Station 3, Mach = 0.5, $Re = 2E6$.

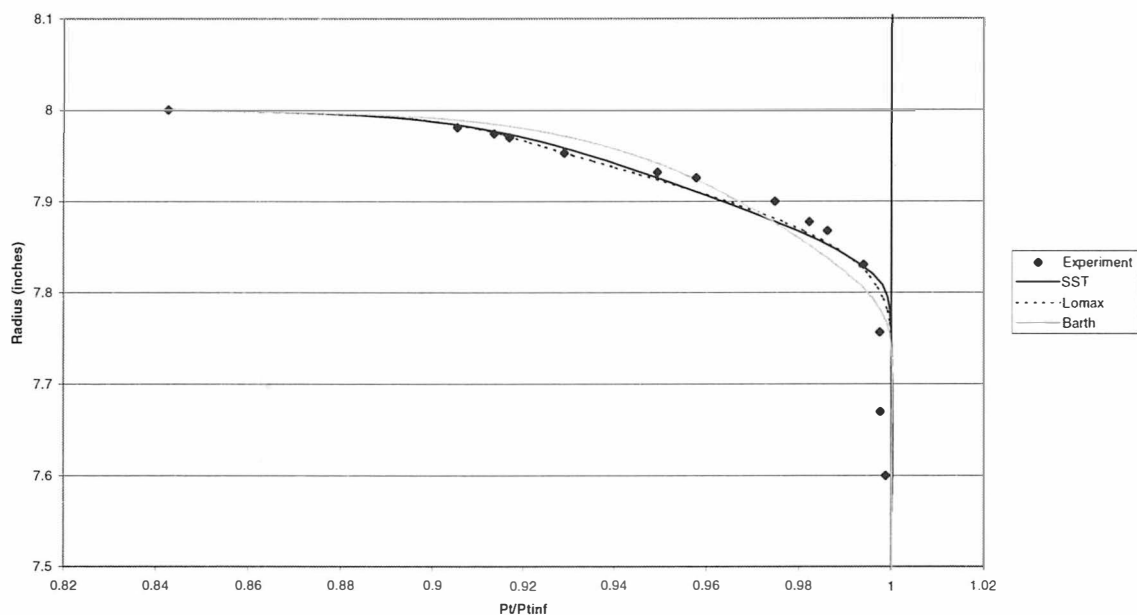


Figure 33. Comparison of Turbulence Model Results for Total Pressure Profile to Experimental Data at Station 3, Mach = 0.5, Re = 3E6.

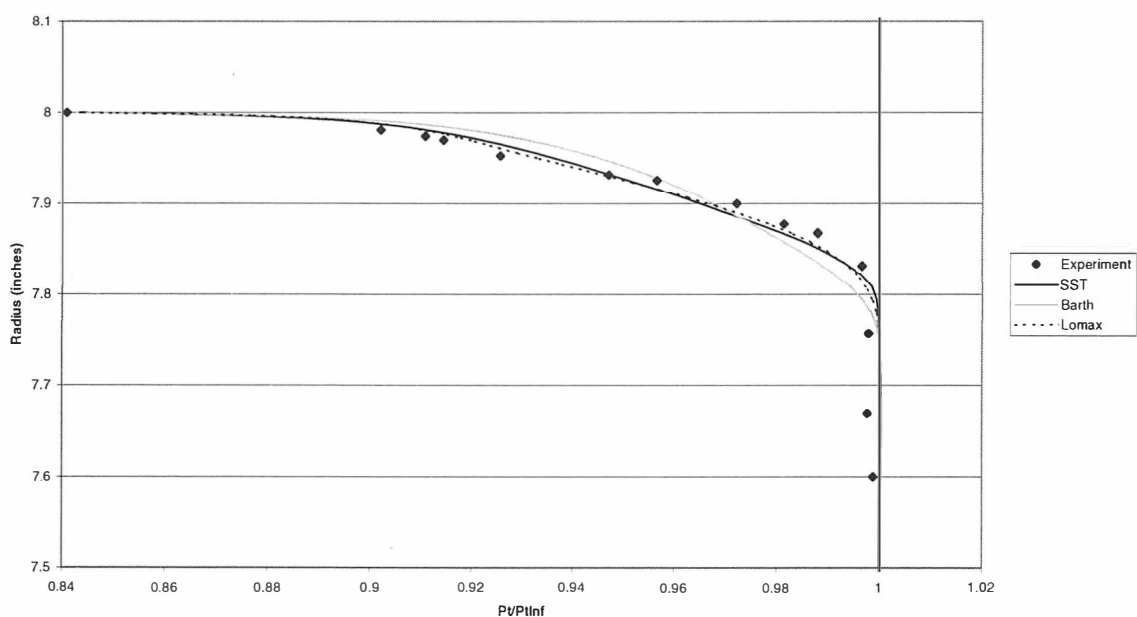


Figure 34. Comparison of Turbulence Model Results for Total Pressure Profile to Experimental Data at Station 3, Mach = 0.5, Re = 3.9E6.

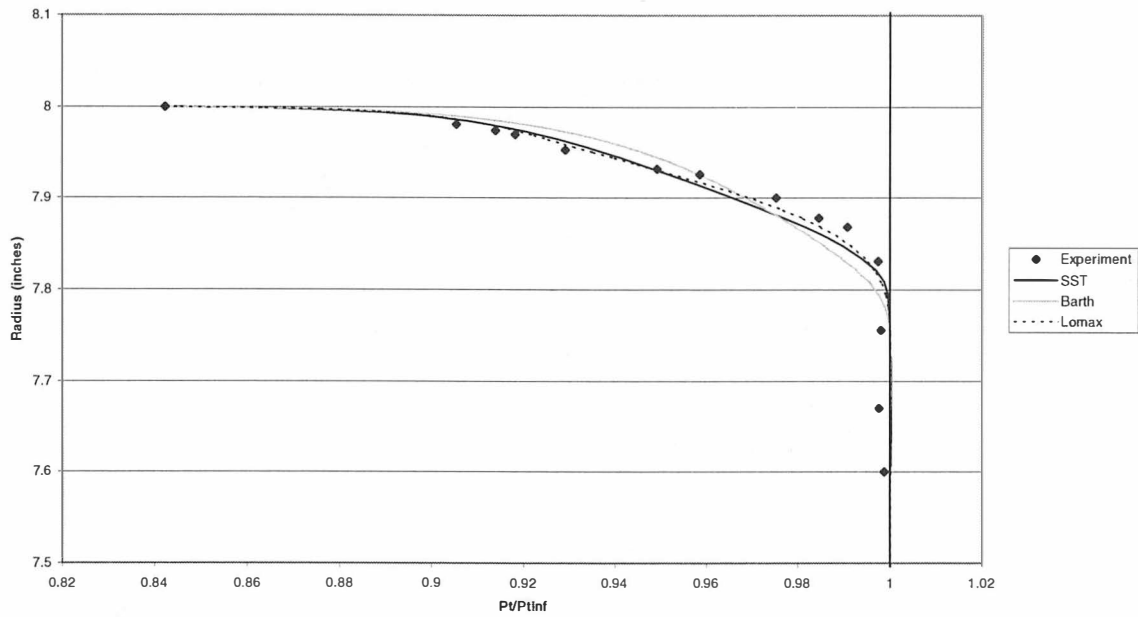


Figure 35. Comparison of Turbulence Model Results for Total Pressure Profile to Experimental Data at Station 3, Mach = 0.5, Re = 5.75E6.

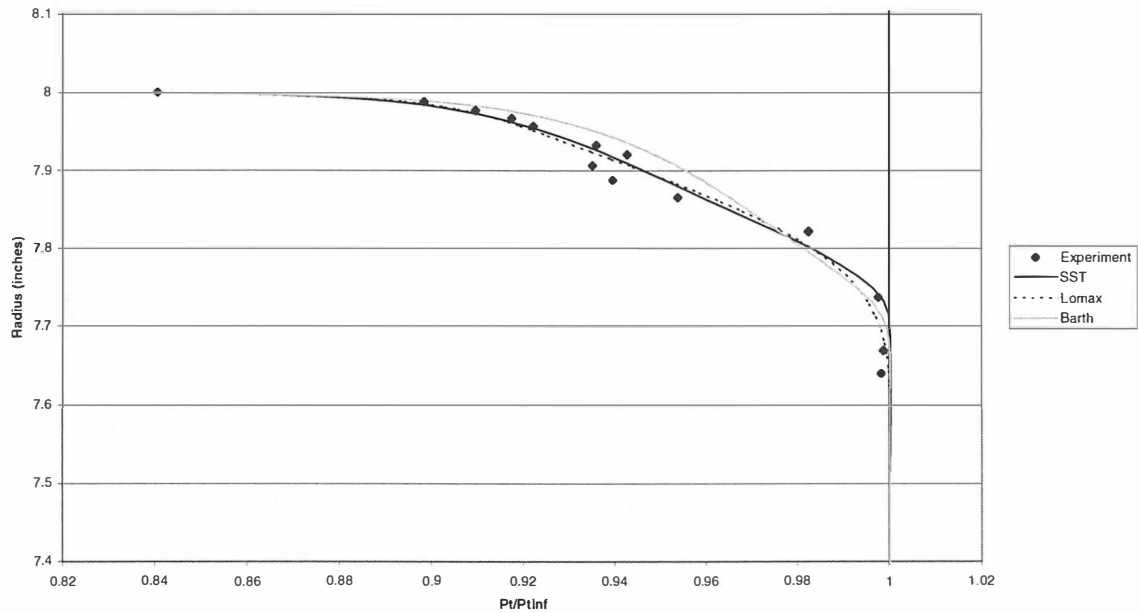


Figure 36. Comparison of Turbulence Model Results for Total Pressure Profile to Experimental Data at Station 8, Mach=0.5, Re = 2E6.

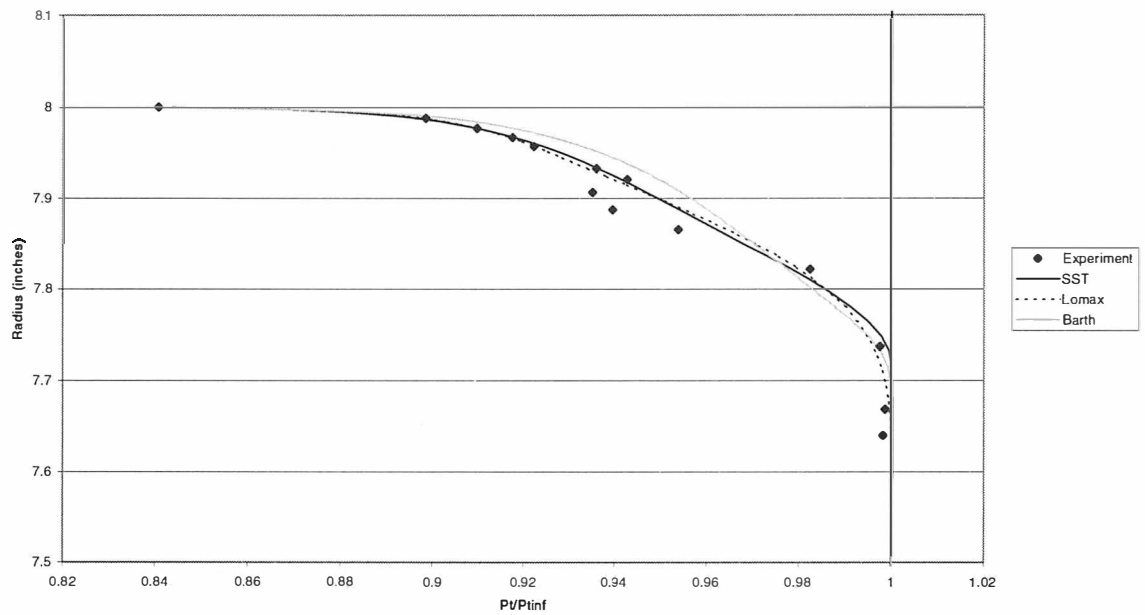


Figure 37. Comparison of Turbulence Model Results for Total Pressure Profiles to Experimental Data at Station 8, Mach = 0.5, Re = 3E6.

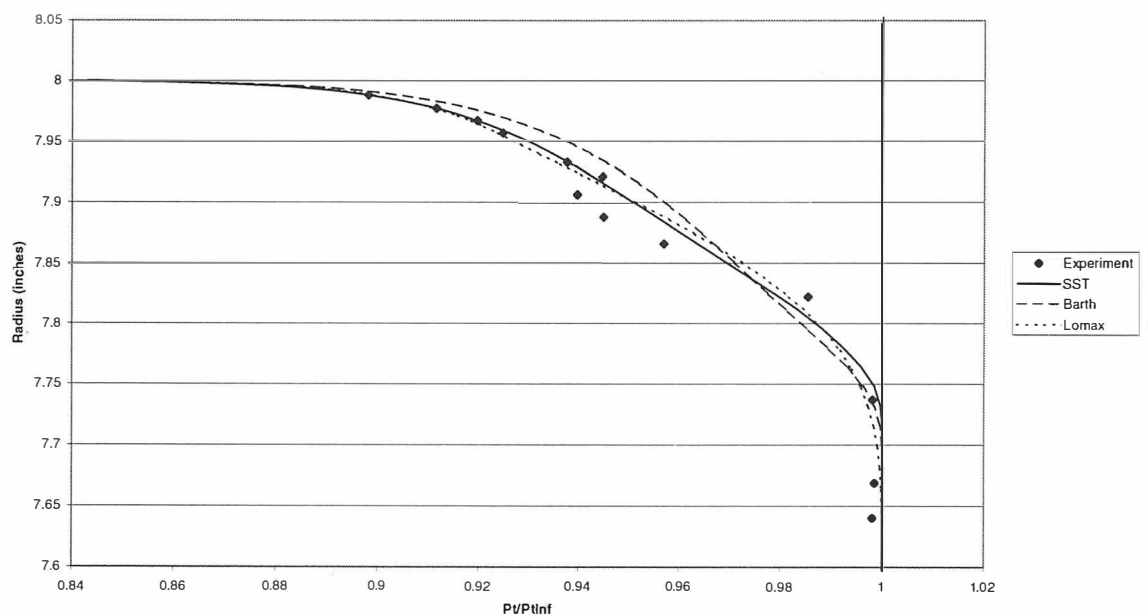


Figure 38. Comparison of Turbulence Model Results for Total Pressure Profiles to Experimental Data at Station 8, Mach = 0.5, Re = 3.9E6.

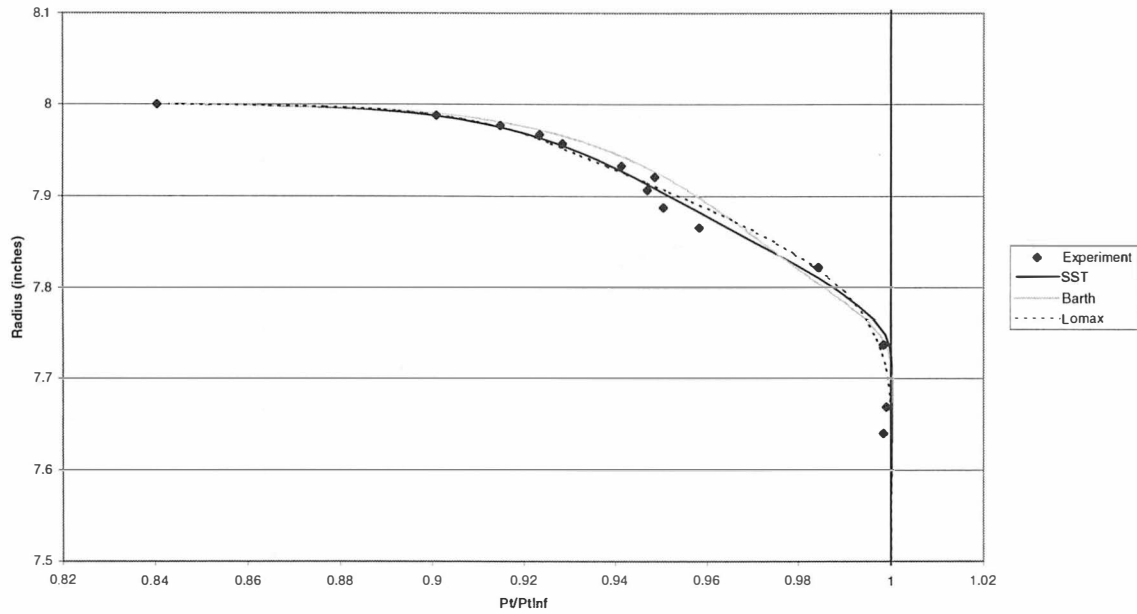


Figure 39. Comparison of Turbulence Model Results for Total Pressure Profiles to Experimental Data at Station 8, Mach 0.5, $Re = 5.75E6$.

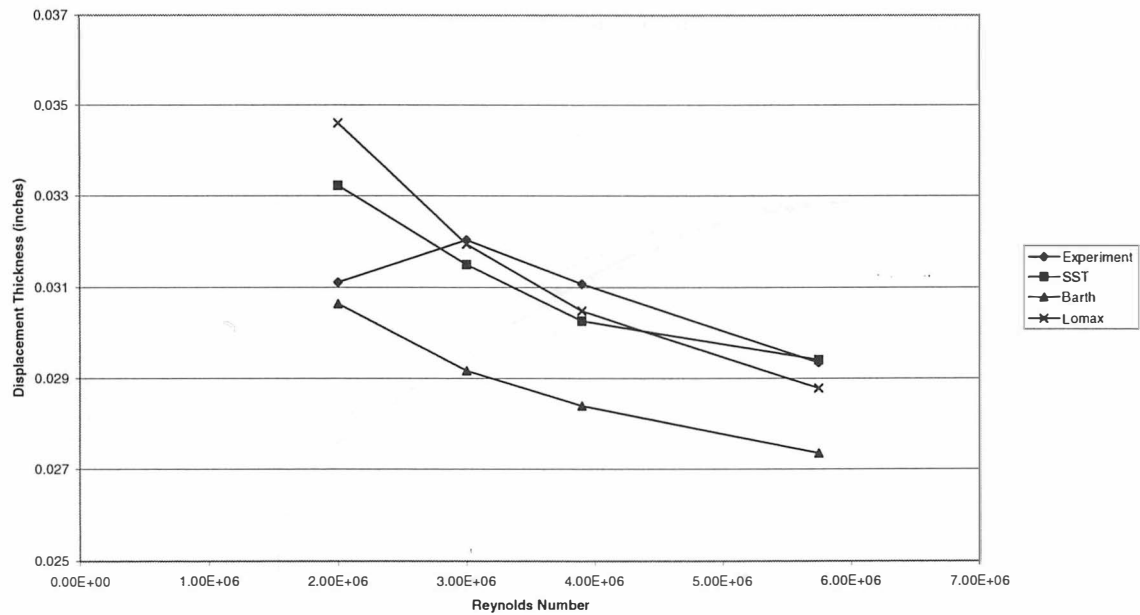


Figure 40. Comparison Between Experimental and Computed Displacement Thickness for Various Turbulence Models at Station 3.

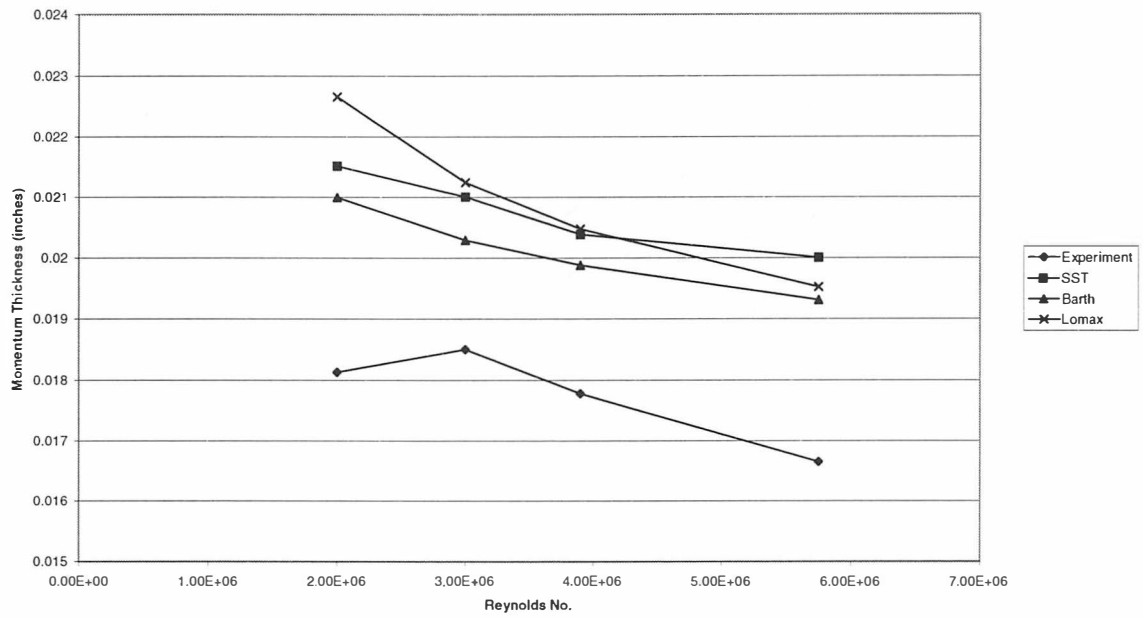


Figure 41. Comparison of Experimental and Computed Momentum Thickness for Various Turbulence Models at Station 3.

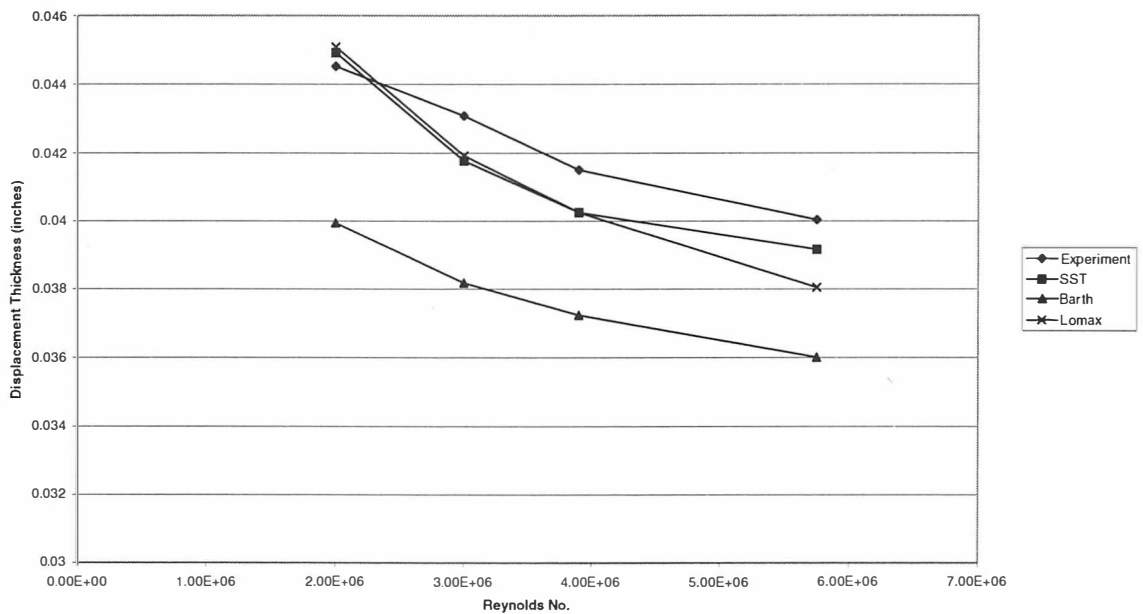


Figure 42. Comparison of Experimental and Computed Displacement Thickness for Various Turbulence Models at Station 8.

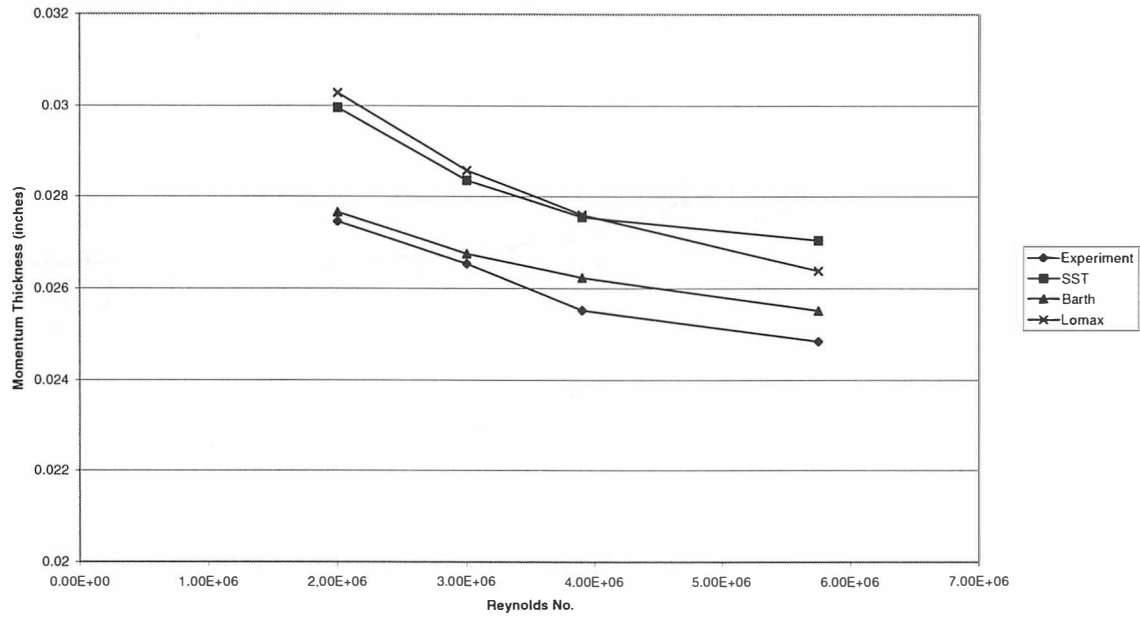


Figure 43. Comparison of Experimental and Computed Momentum Thickness for Various Turbulence Models at Station 8.

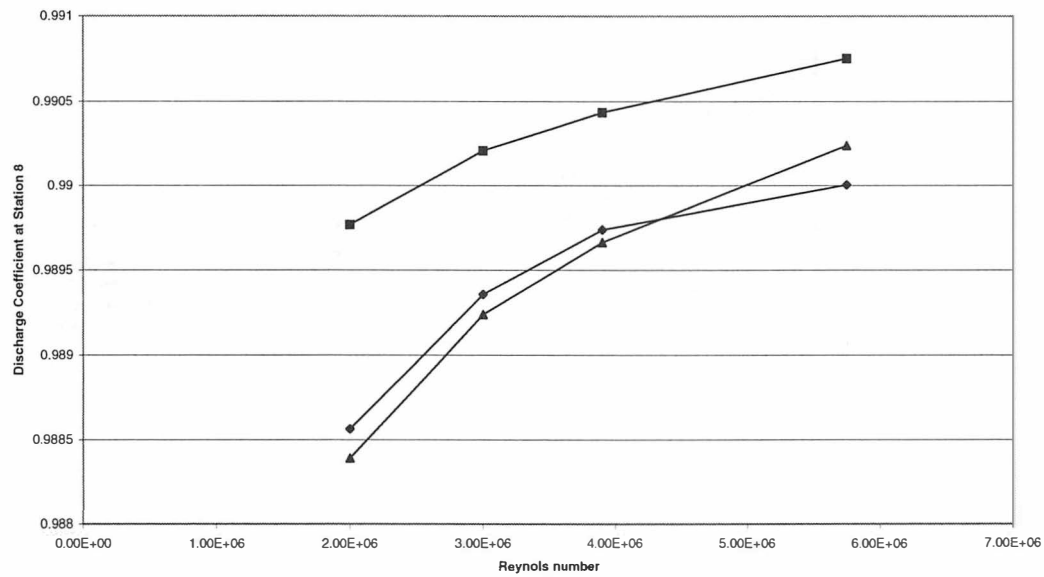


Figure 44. Comparison of Calculated Discharge Coefficient at Station 8 for Various Turbulence Models.

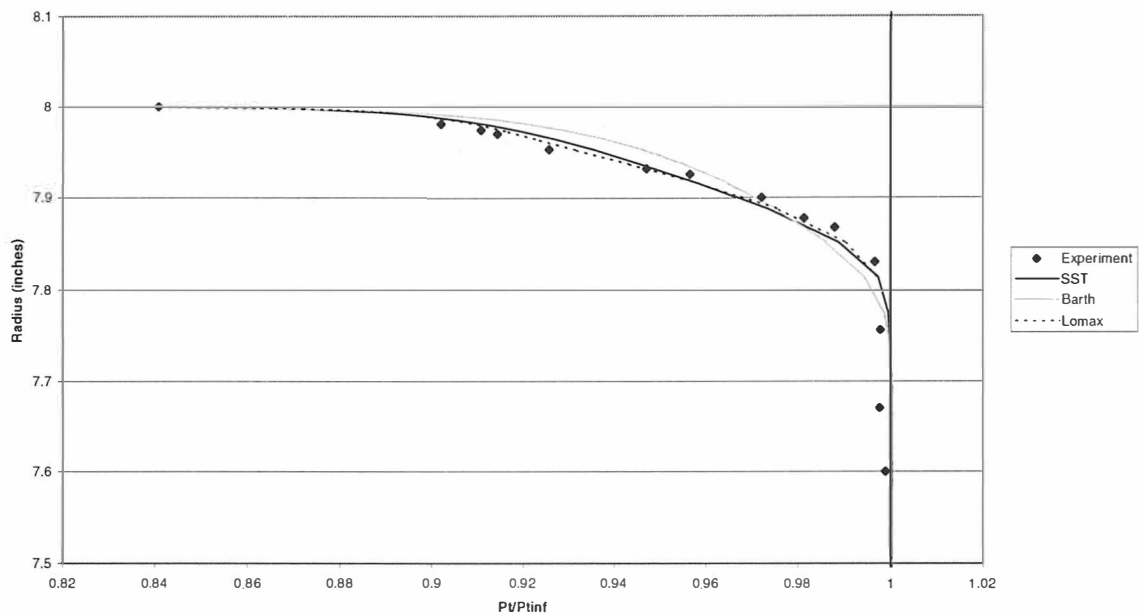


Figure 45. Comparison of Turbulence Model Results for Total Pressure Profiles at Half Nominal Grid Resolution: Station 3, $M=0.5$, $Re=3.9E6$.

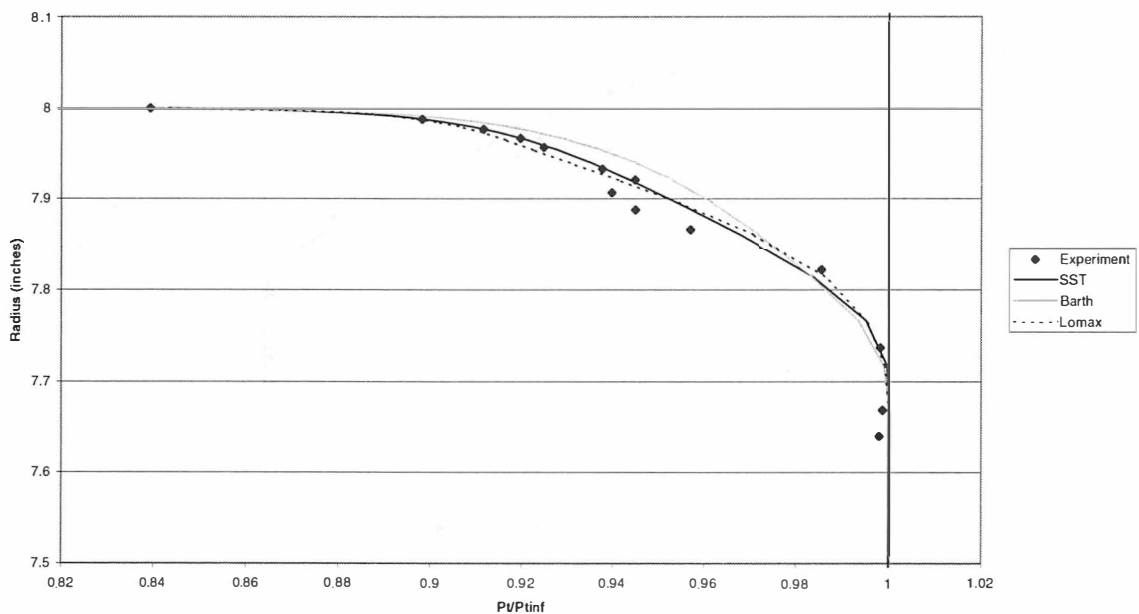


Figure 46. Comparison of Turbulence Model Results for Total Pressure Profiles at Half Nominal Grid Resolution: Station 8, $M=0.5$, $Re=3.9E6$.

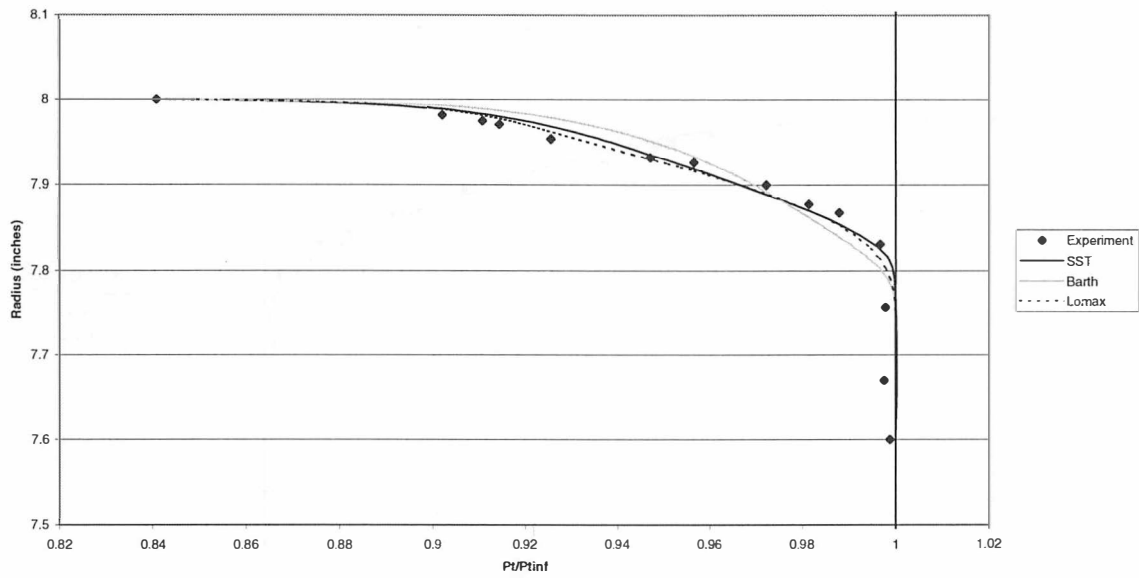


Figure 47. Comparison of Turbulence Model Results for Total Pressure Profiles at Double Nominal Grid Resolution: Station 3, $M=0.5$, $Re=3.9E6$.

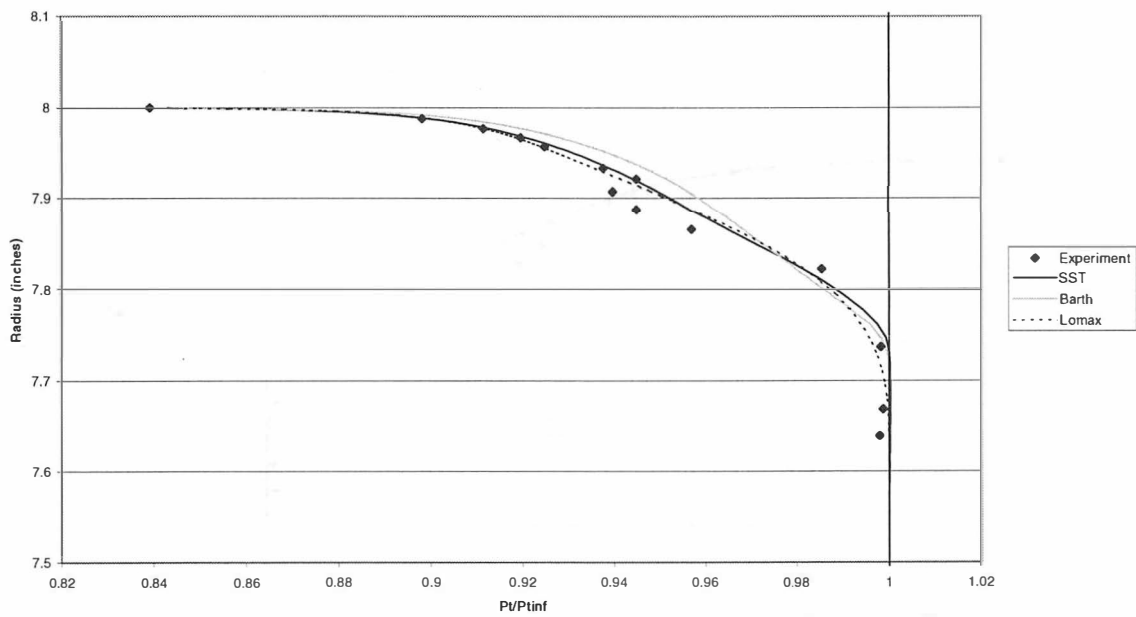


Figure 48. Comparison of Turbulence Model Results for Total Pressure Profiles at Double Nominal Grid Resolution: Station 8, $M=0.5$, $Re=3.9E6$.

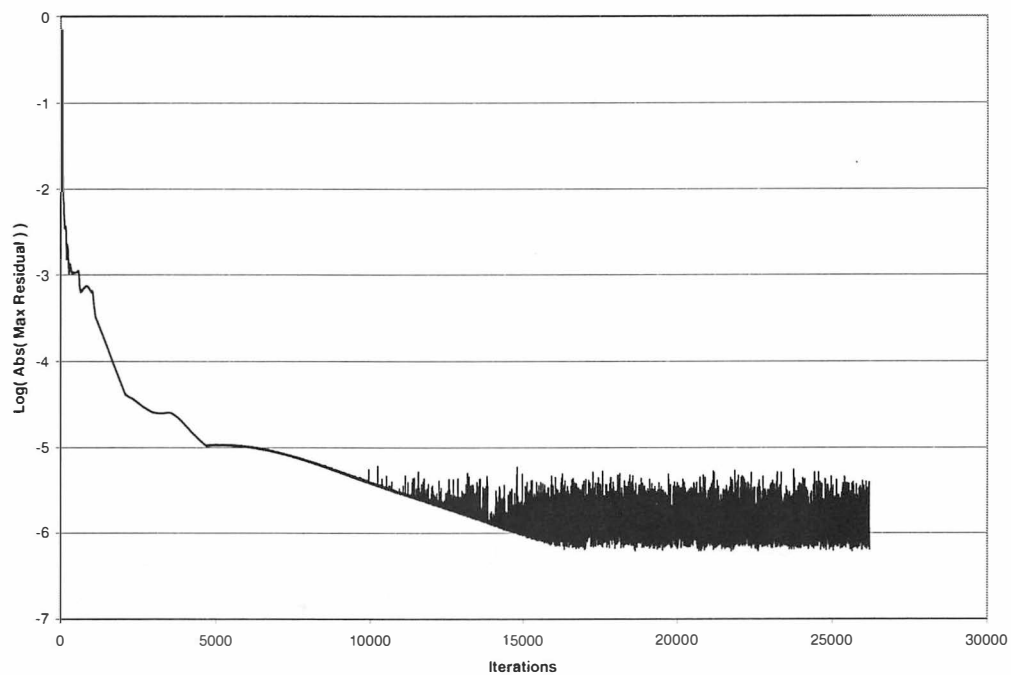


Figure 49. Max Residual, $M=0.5$, $Re=5.75E6$.

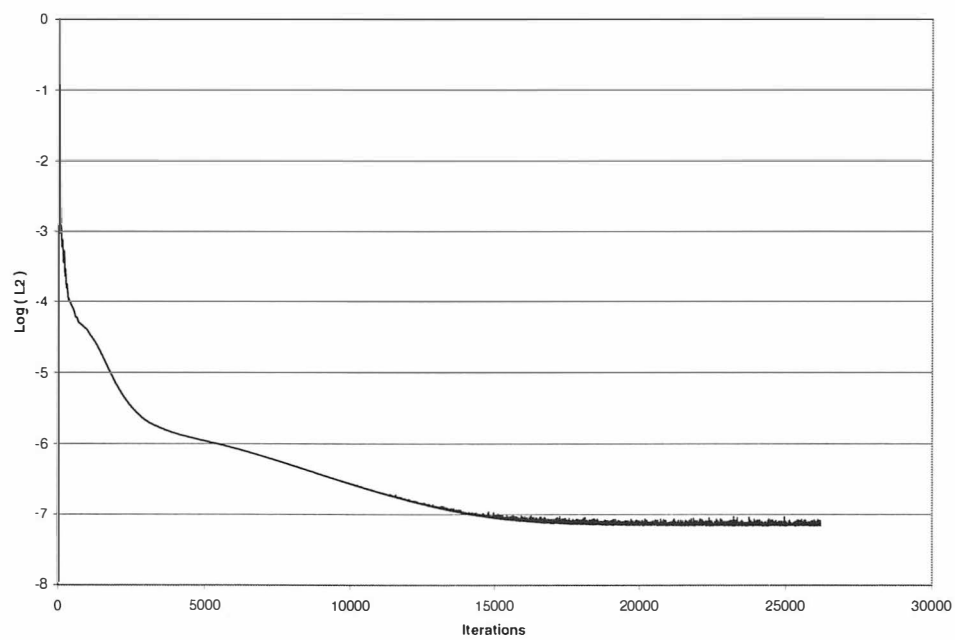


Figure 50. L2-norm Residual, $M=0.5$, $Re=5.75E6$.

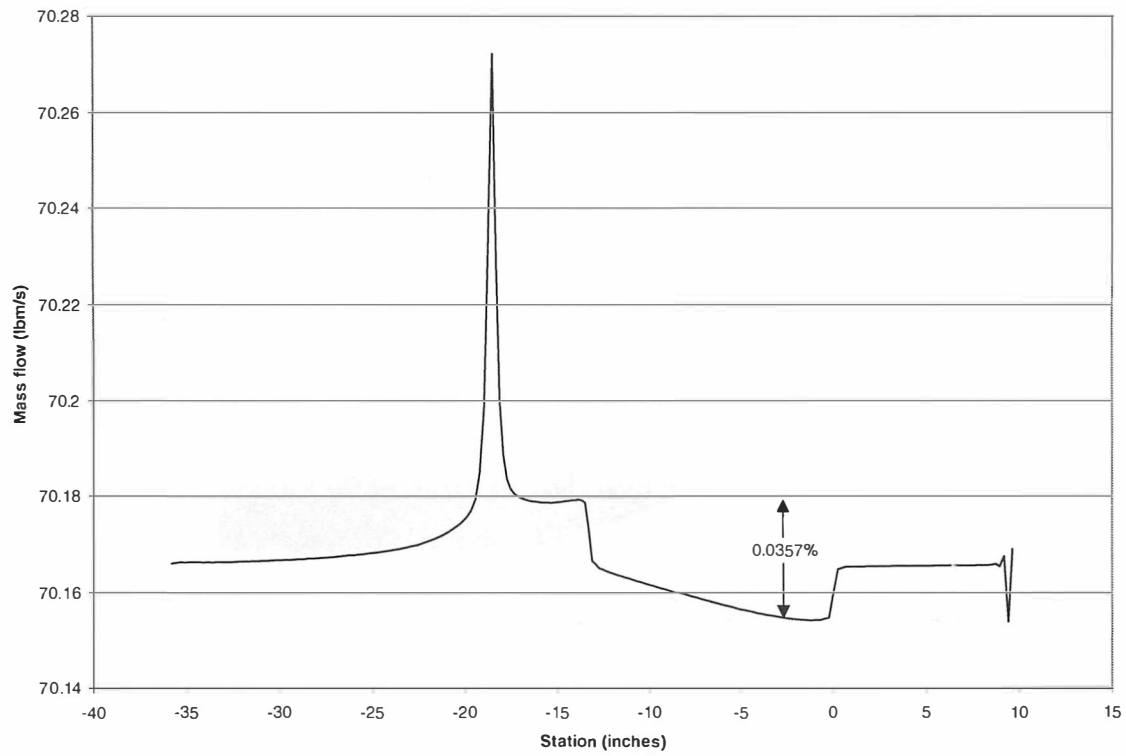


Figure 51. Calculated Mass Flow, $M=0.5$, $Re=5.75E6$.

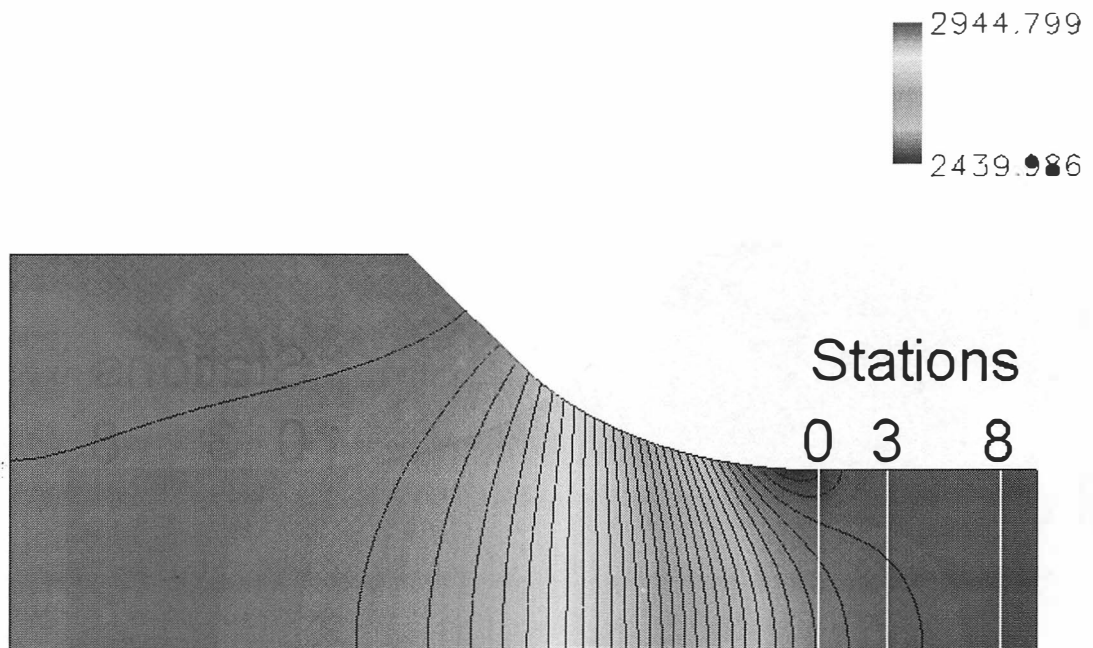


Figure 52. Pressure Contour, Mach = 0.5, Re = 5.75E6.

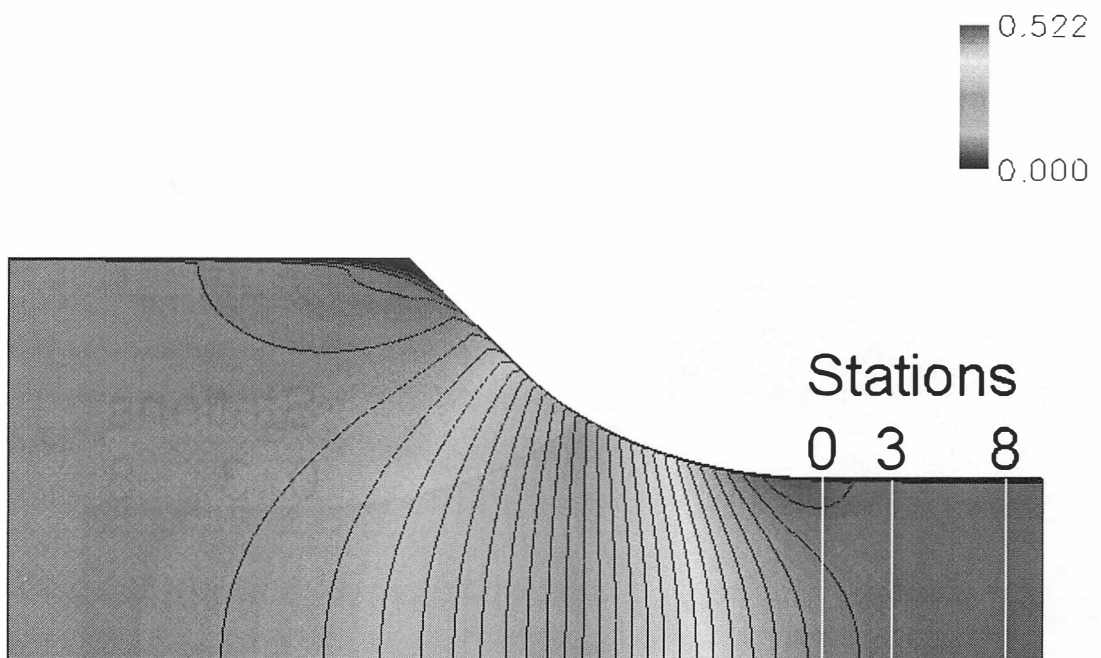
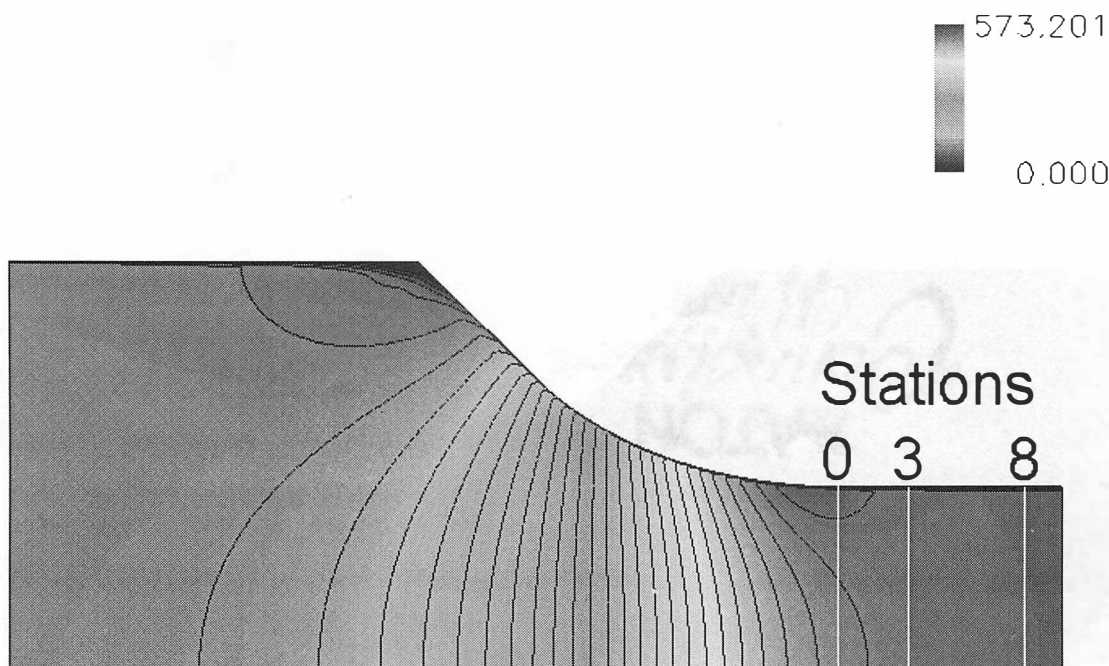


Figure 53. Mach Contour, Mach = 0.5, Re = 5.75E6.



Figures 54. Velocity Contour, Mach = 0.5, Re = 5.75E6.

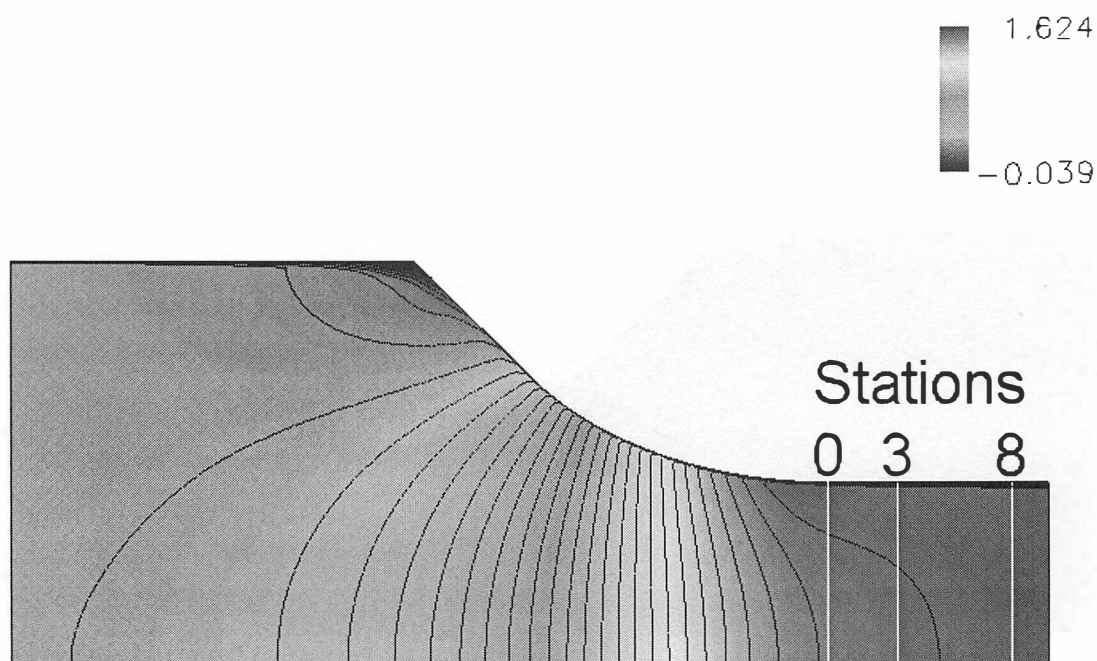


Figure 55. Mass Flux Contours, Mach =0.5, Re = 5.75E6.

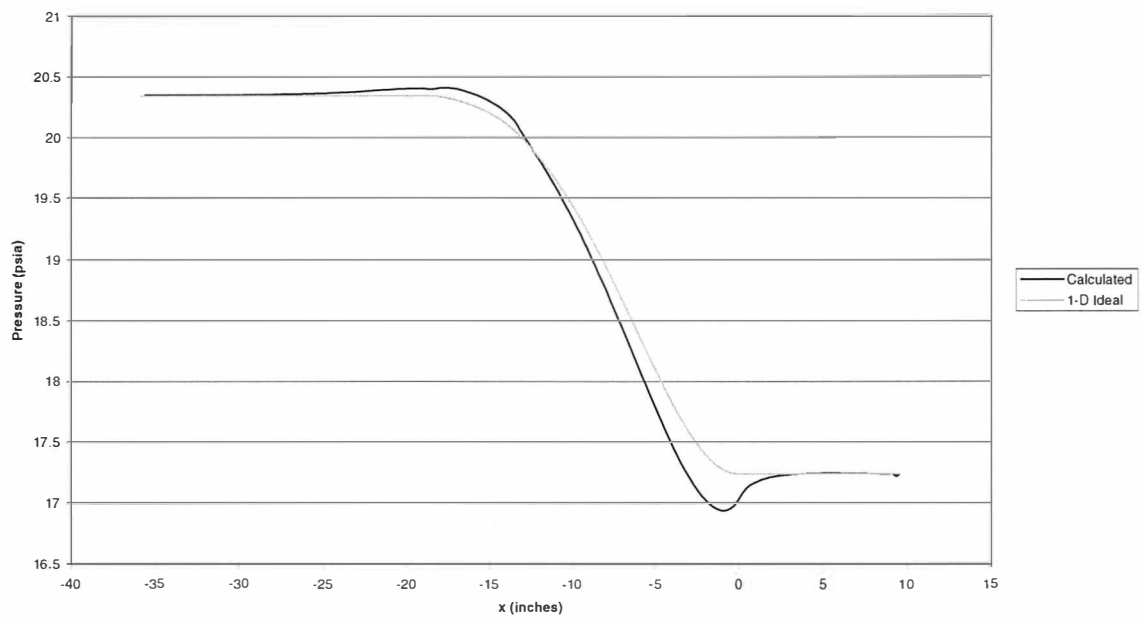


Figure 56. Wall Static Pressure, $M=0.5$, $Re=5.75E6$.

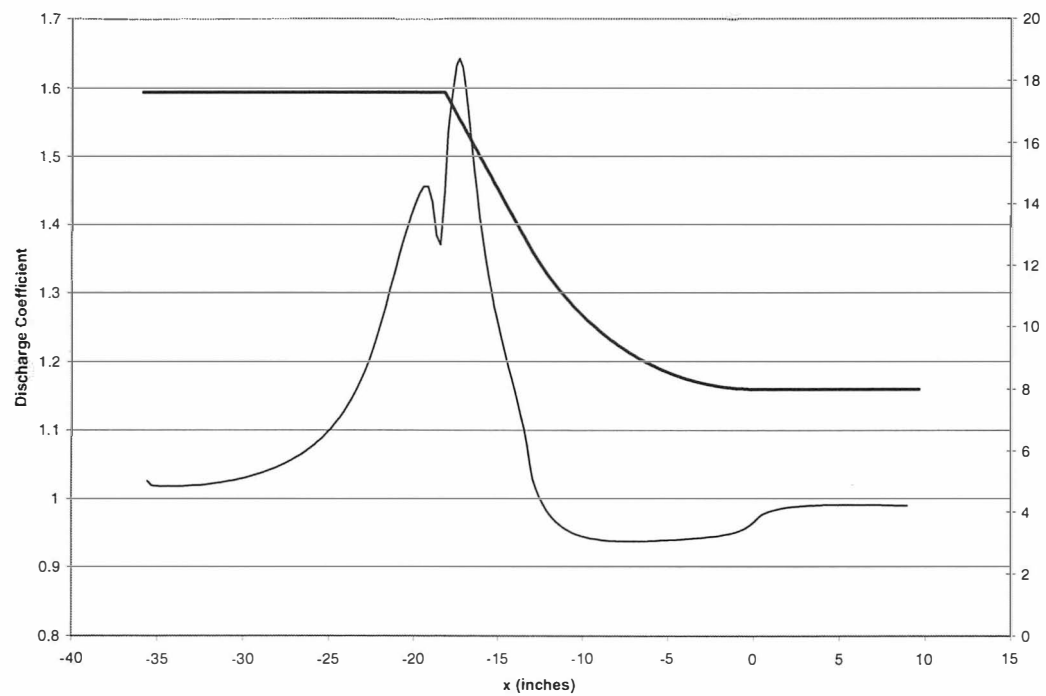


Figure 57. Discharge Coefficient and Wall Profile, $M=0.5$, $Re=5.75E6$.

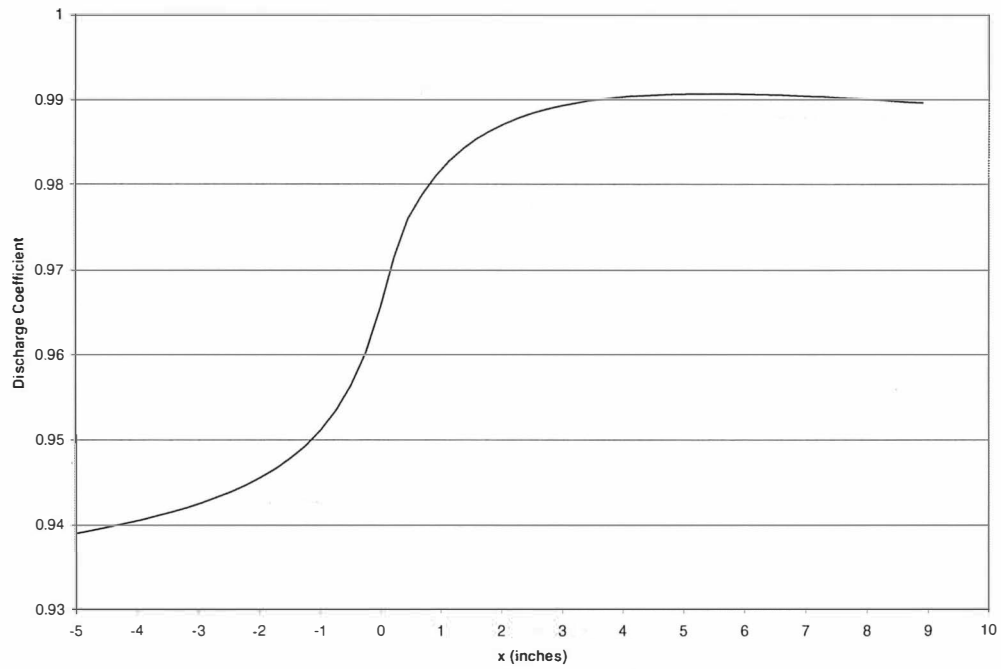


Figure 58. Discharge Coefficient in the Throat Region, Mach = 0.5, Re = 5.75E6.

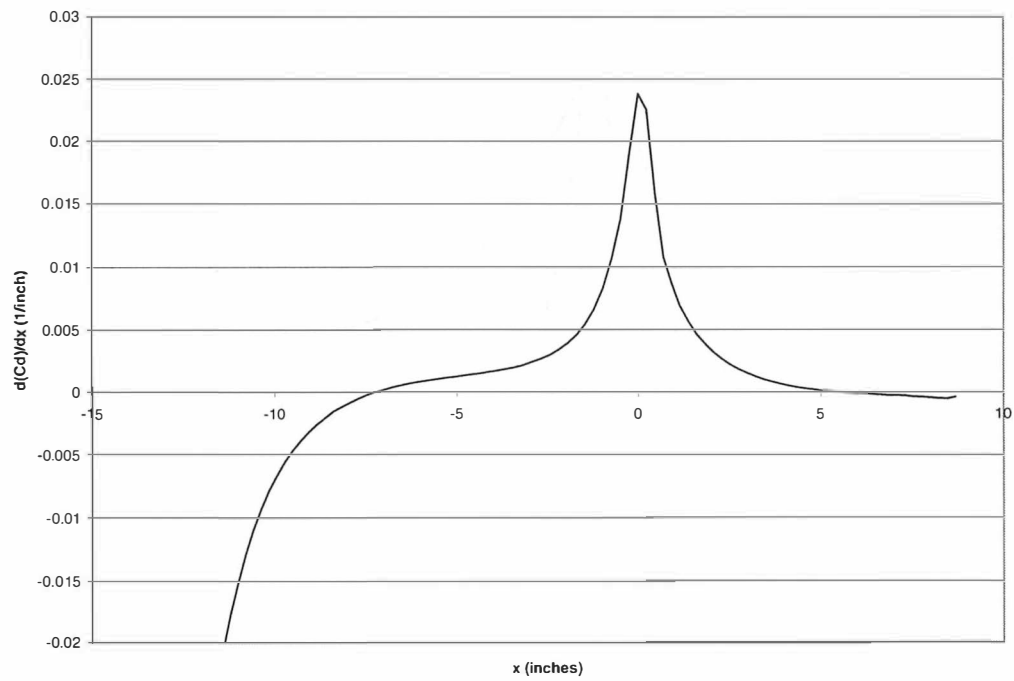


Figure 59. Discharge Coefficient Slope, M=0.5, Re=5.75E6.

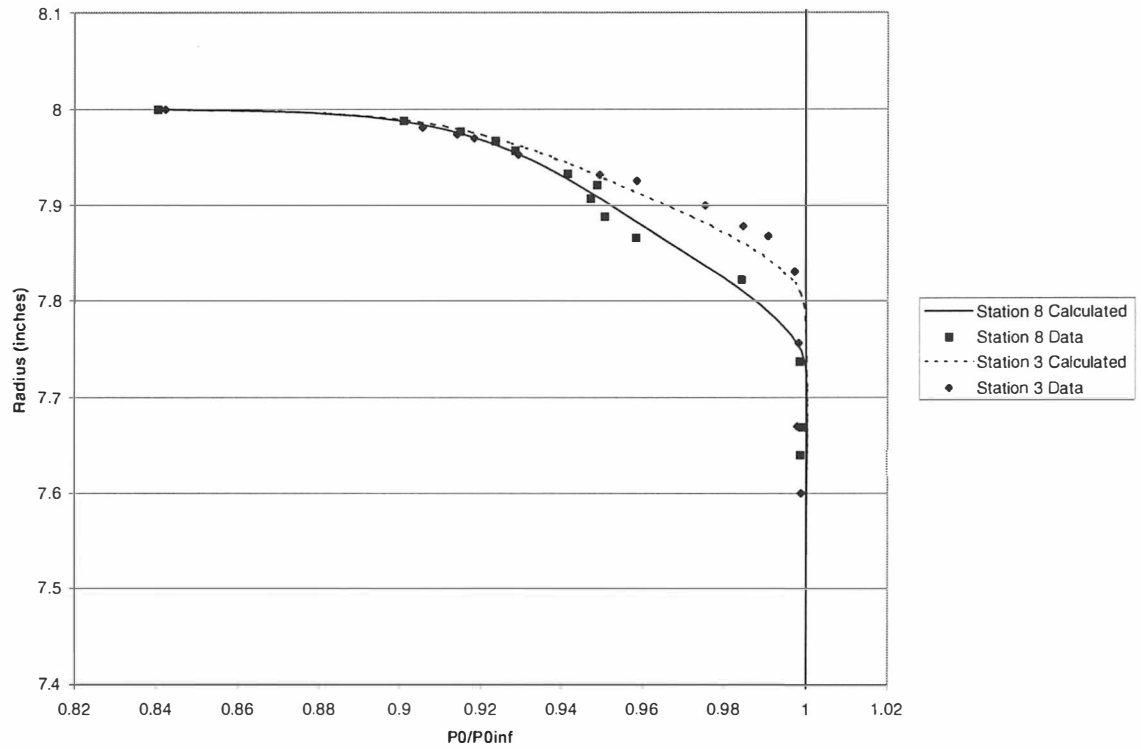


Figure 60. Comparison of Predicted and Experimental Total Pressure Ratio in Boundary Layer, $M=0.5$, $Re=5.75E6$.

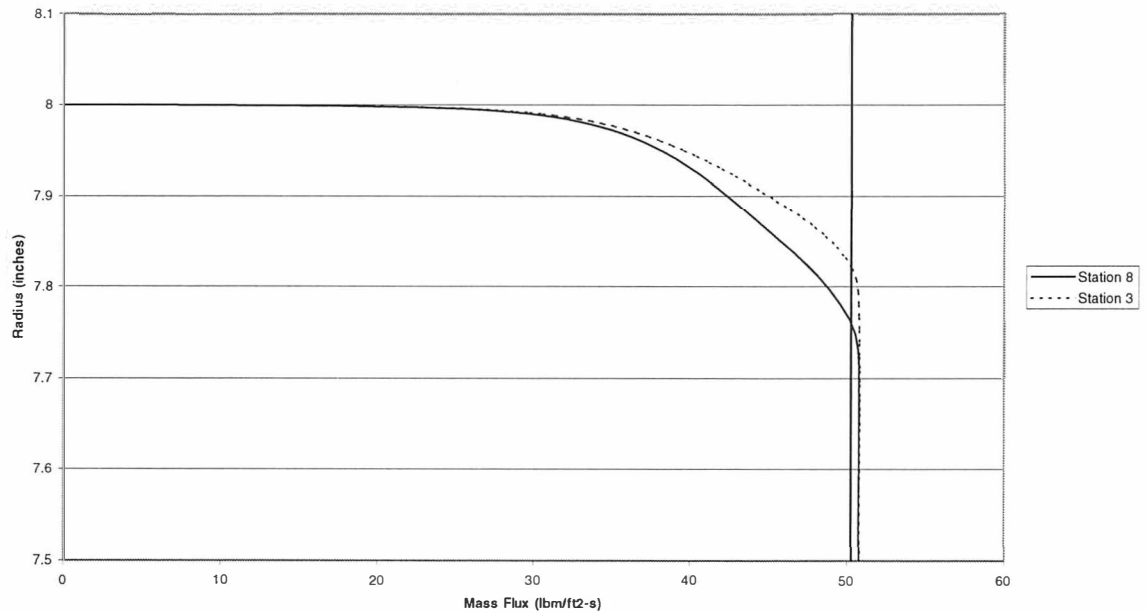


Figure 61. Mass Flux in Boundary Layer, $M=0.5$, $Re=5.75E6$.

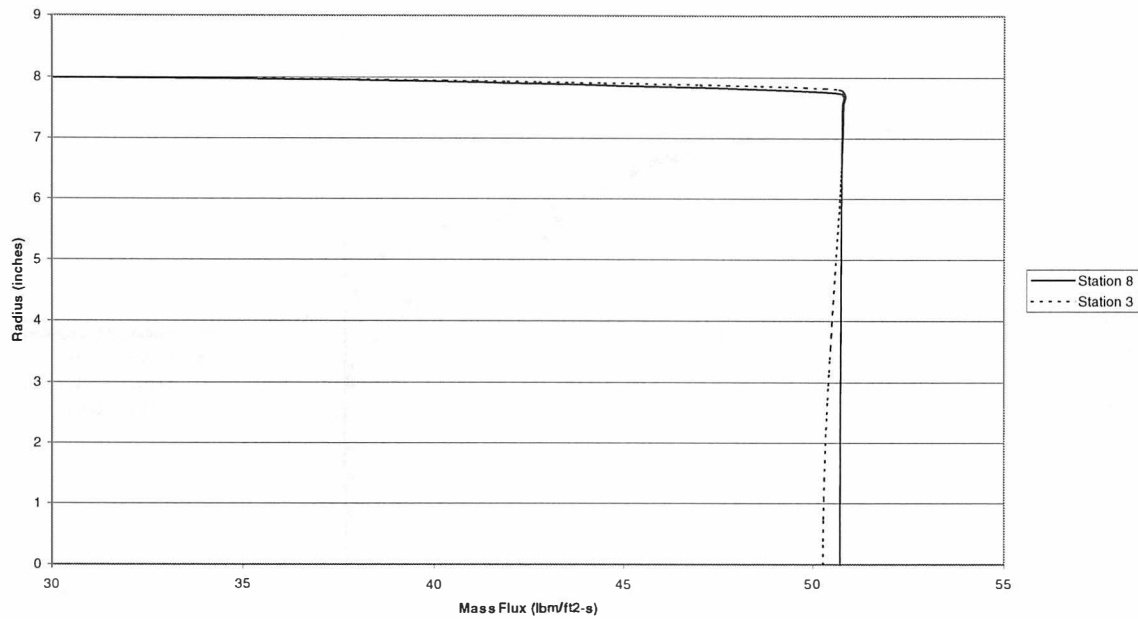


Figure 62. Mass Flux, Mach = 0.5, Re=5.75E6.

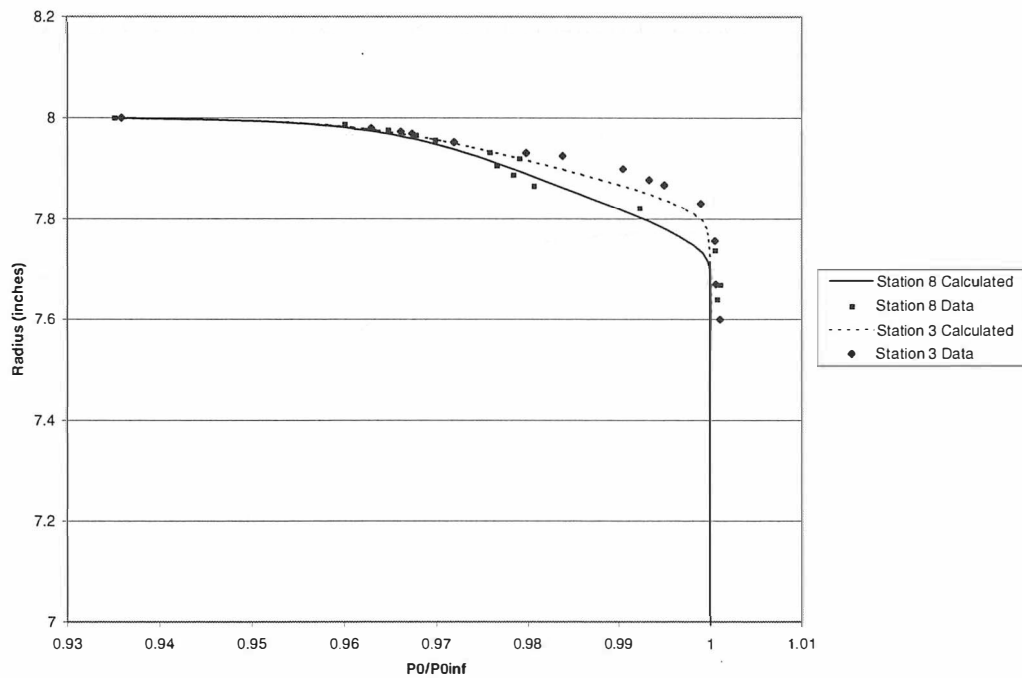


Figure 63. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, M=0.31, Re=2E6.

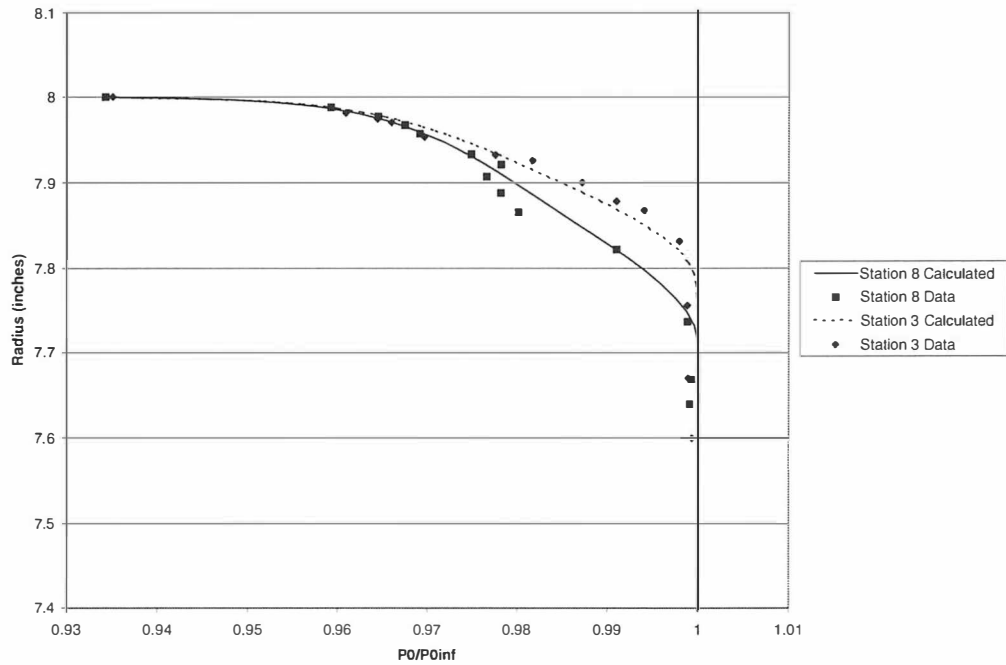


Figure 64. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.31$, $Re=3E6$.

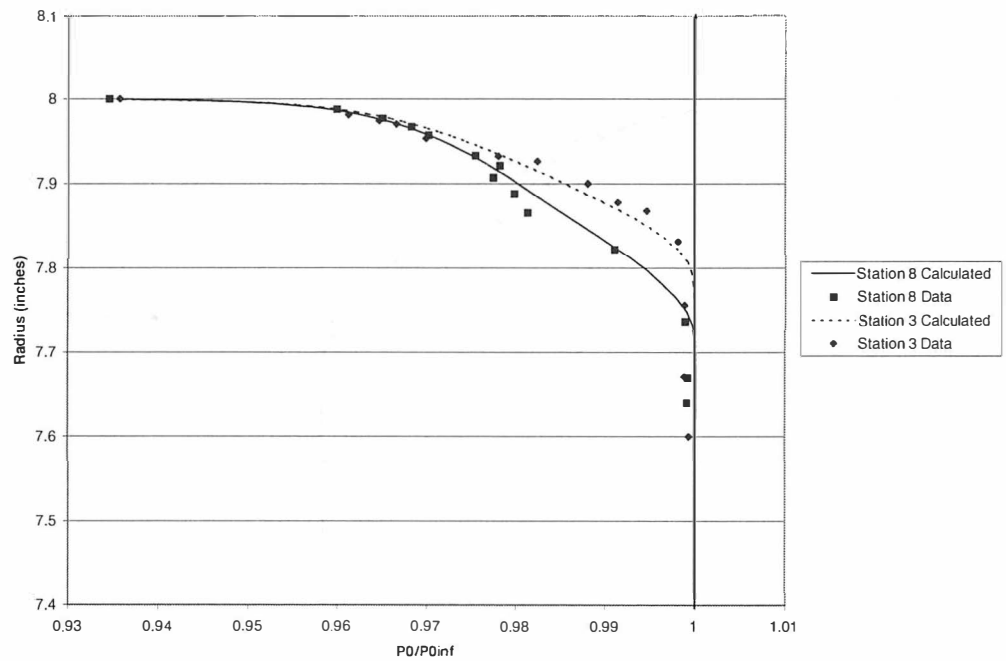


Figure 65. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.31$, $Re=3.9E6$.

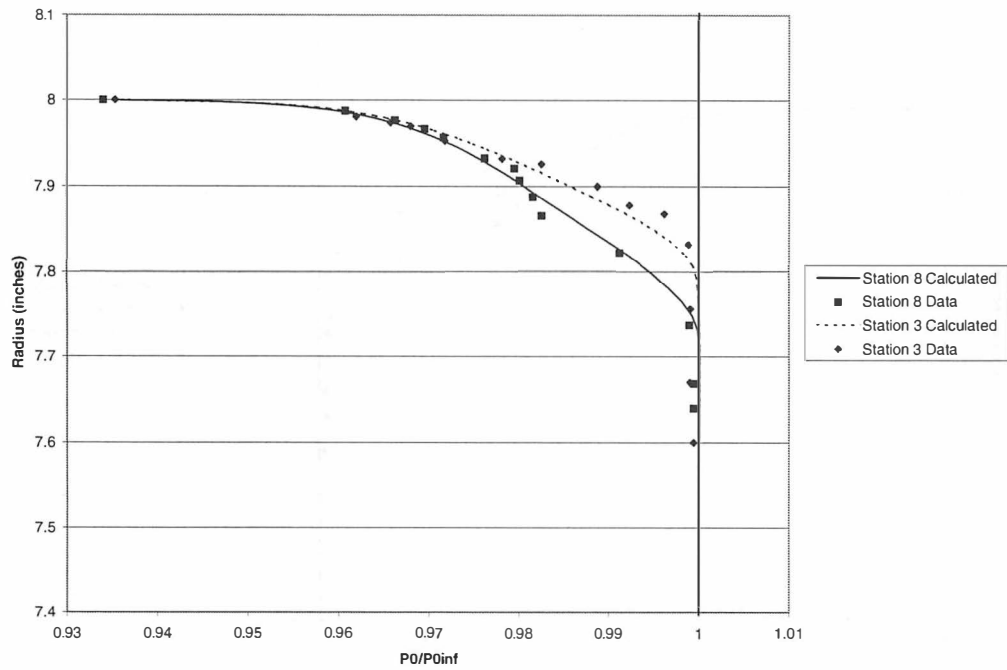


Figure 66. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.31$, $Re=5.75E6$.

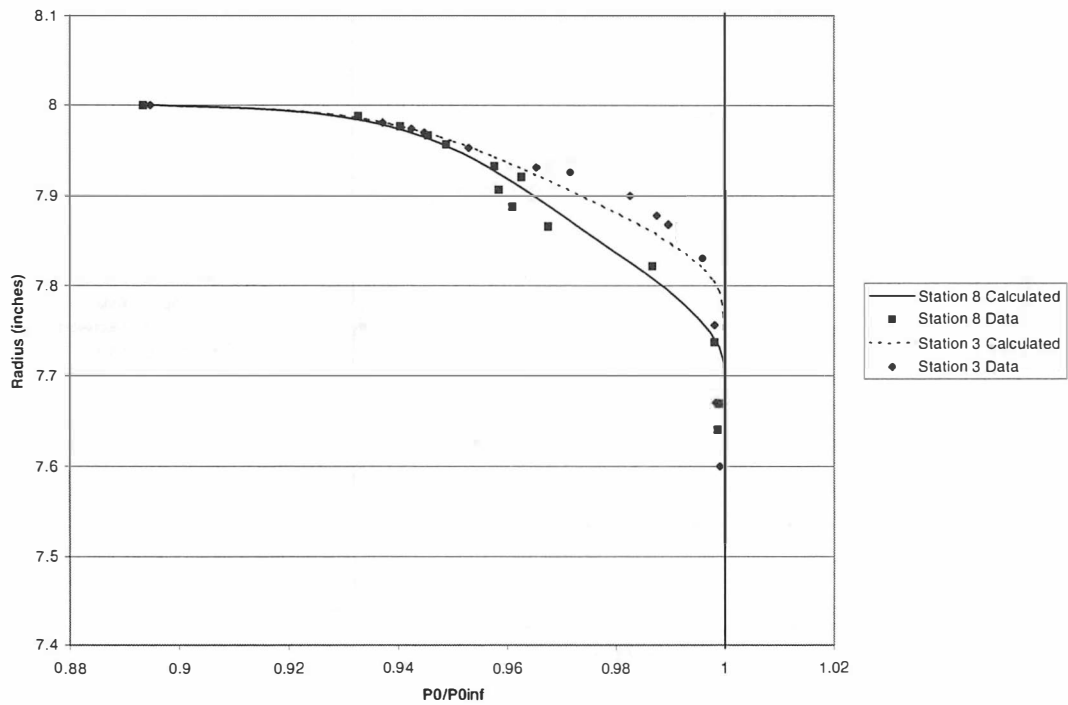


Figure 67. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.4$, $Re=2E6$.

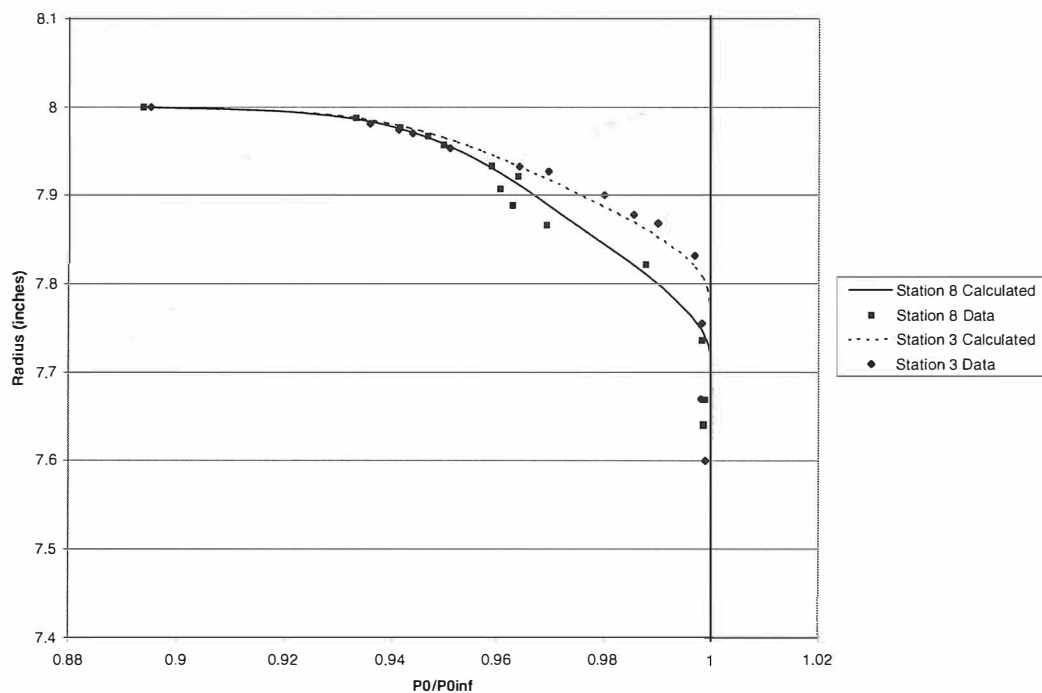


Figure 68. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.4$, $Re=3E6$.

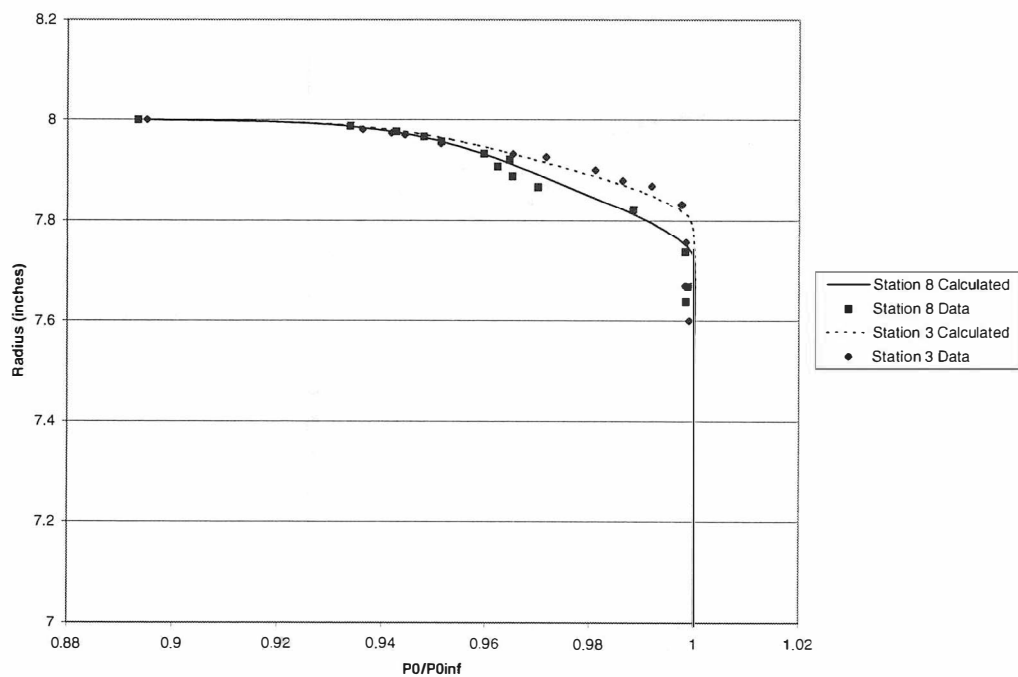


Figure 69. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M = 0.4$, $Re=3.9E6$.

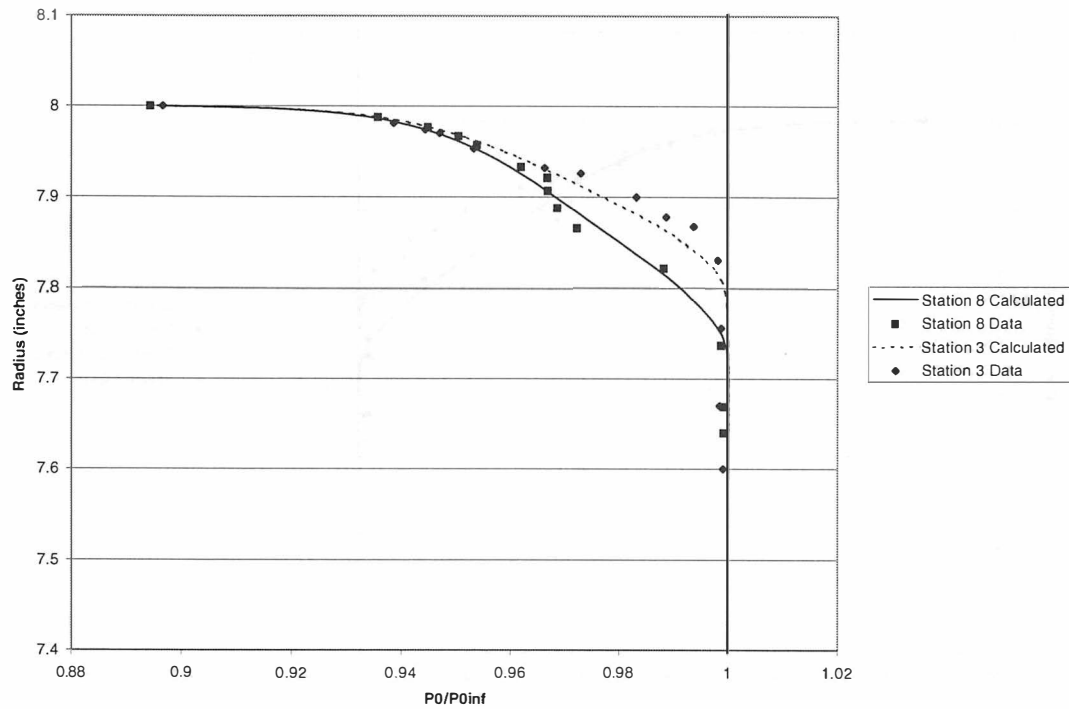


Figure 70. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.4$, $Re=5.75E6$.

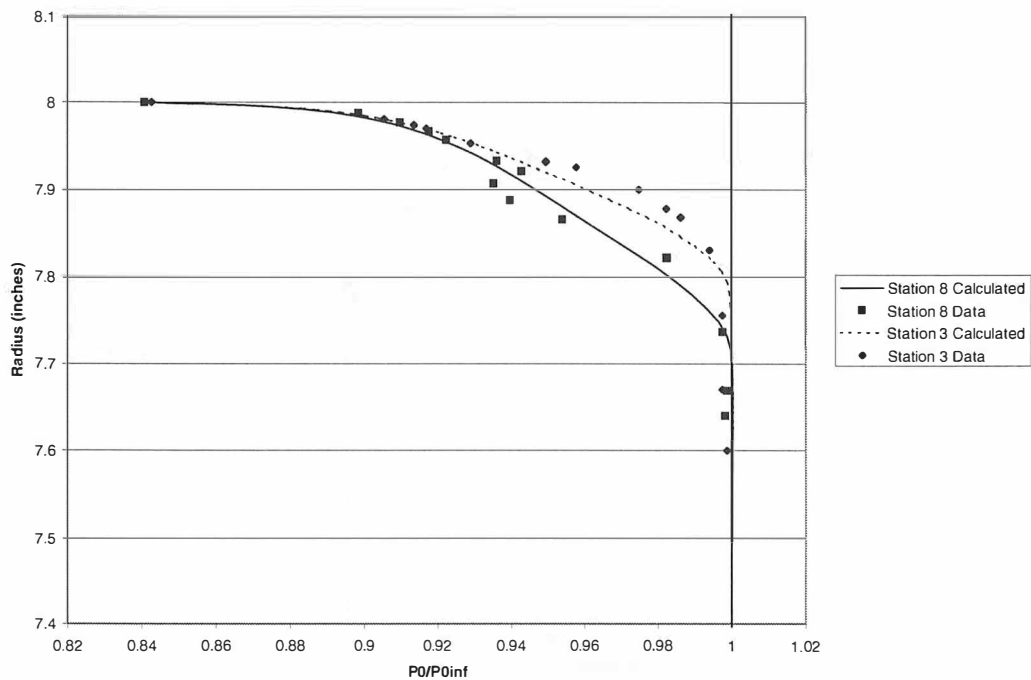


Figure 71. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.5$, $Re=2E6$.

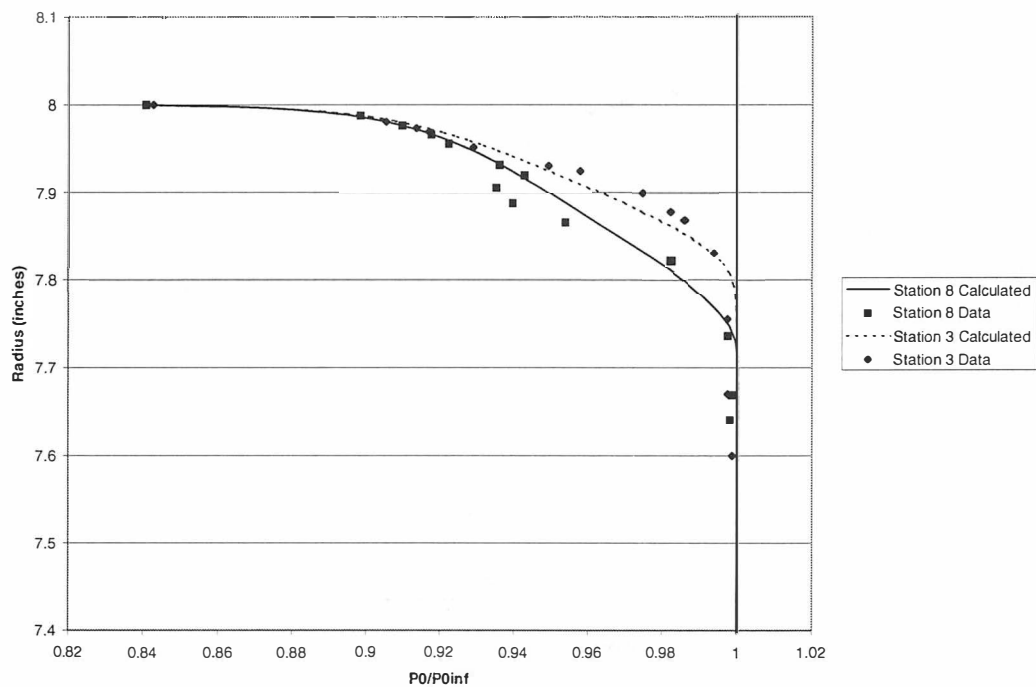


Figure 72. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.5$, $Re=3E6$.

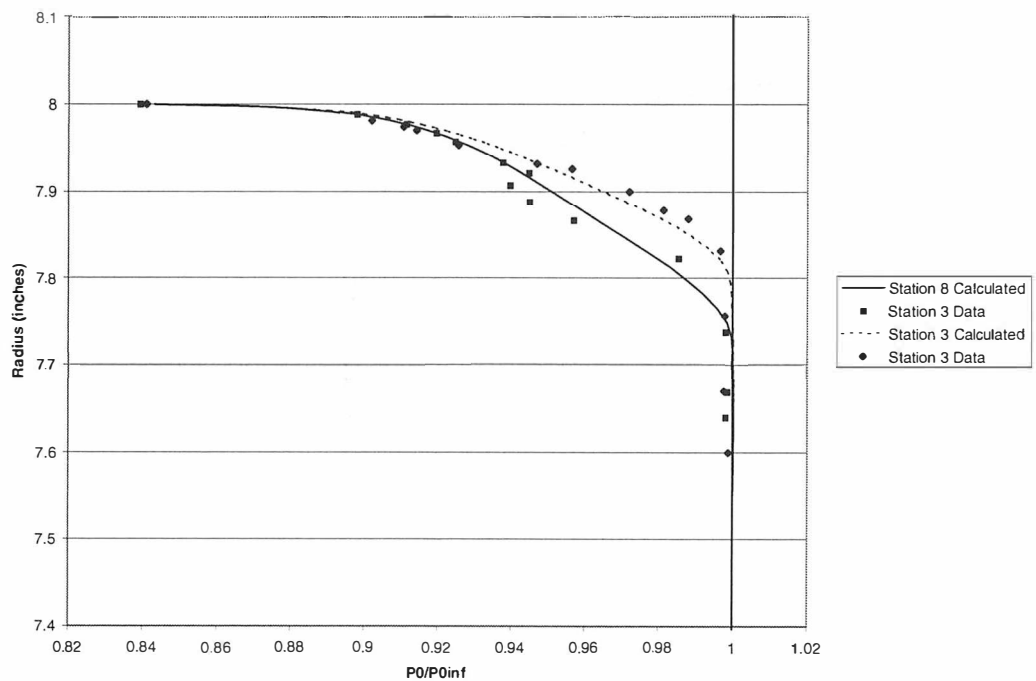


Figure 73. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.5$, $Re=3.9E6$.

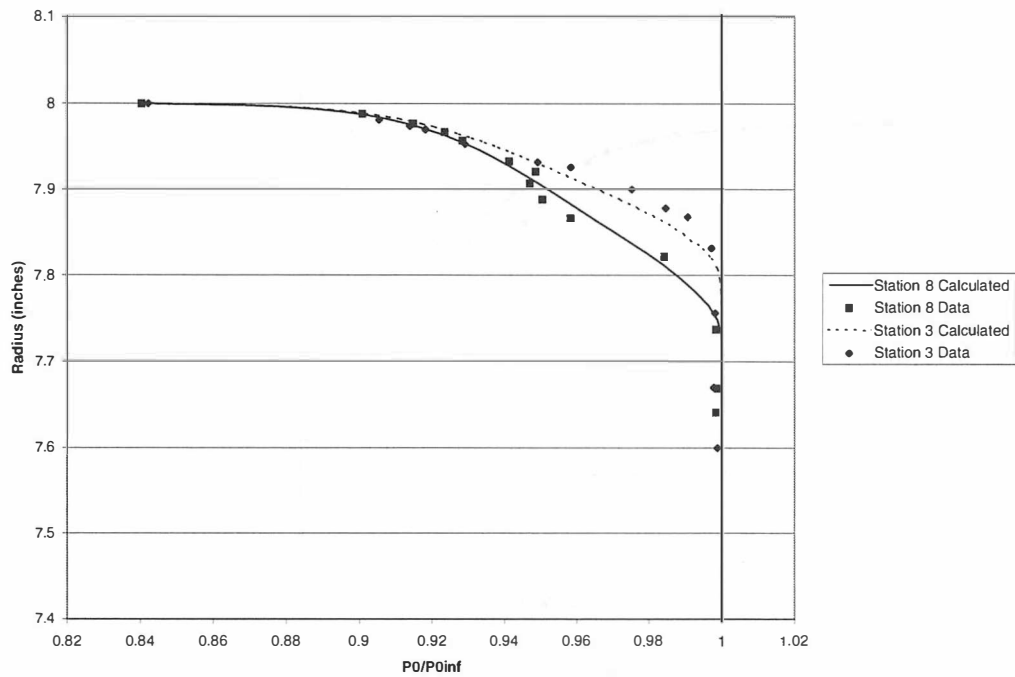


Figure 74. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.5$, $Re=5.75E6$.

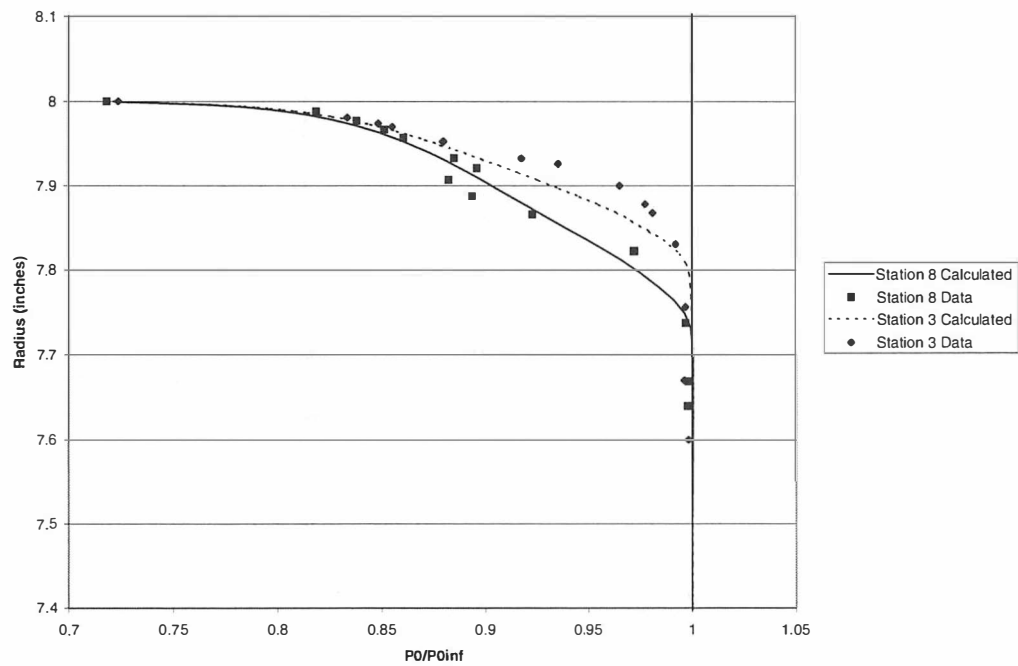


Figure 75. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.7$, $Re=2E6$.

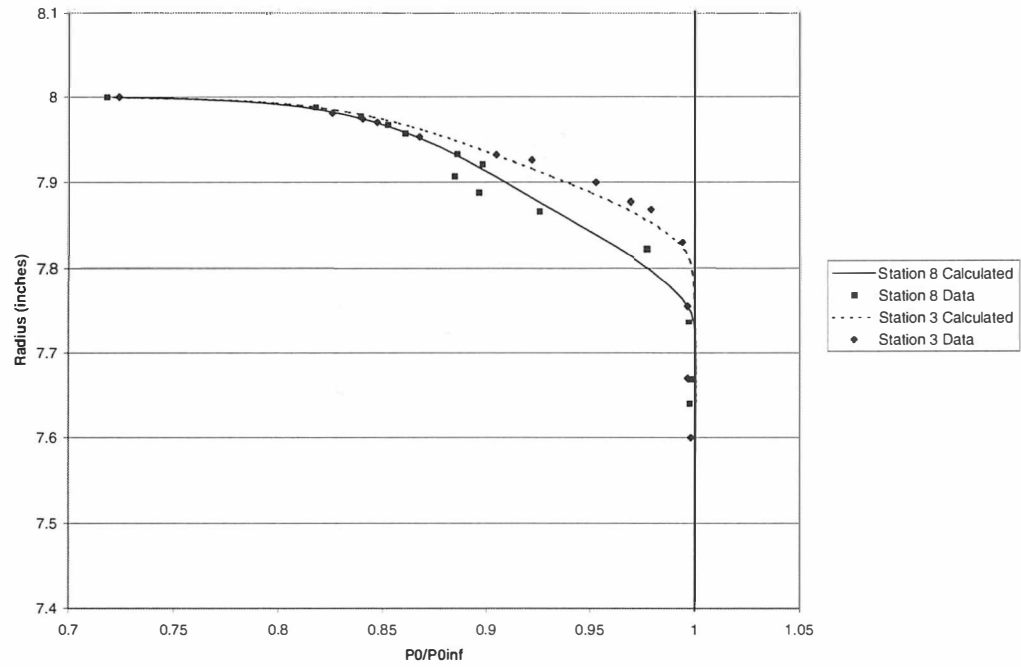


Figure 76. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.7$, $Re=3E6$.

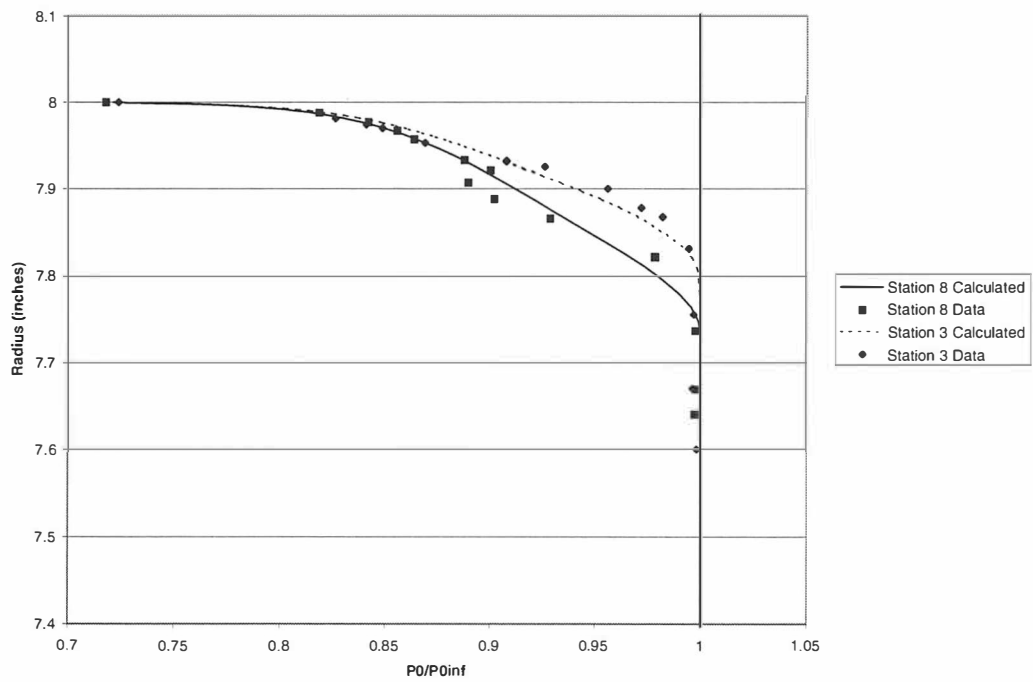


Figure 77. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.7$, $Re=3.9E6$.

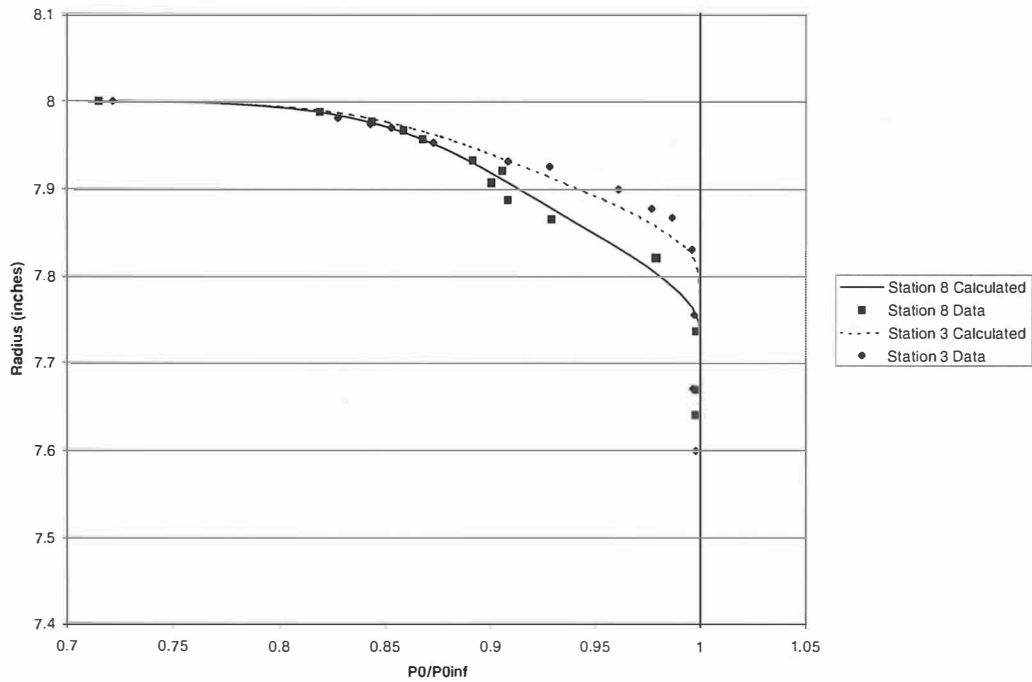


Figure 78. Comparison of Predicted and Experimental Boundary Layer Total Pressure Profiles, $M=0.7$, $Re=5.75E6$.

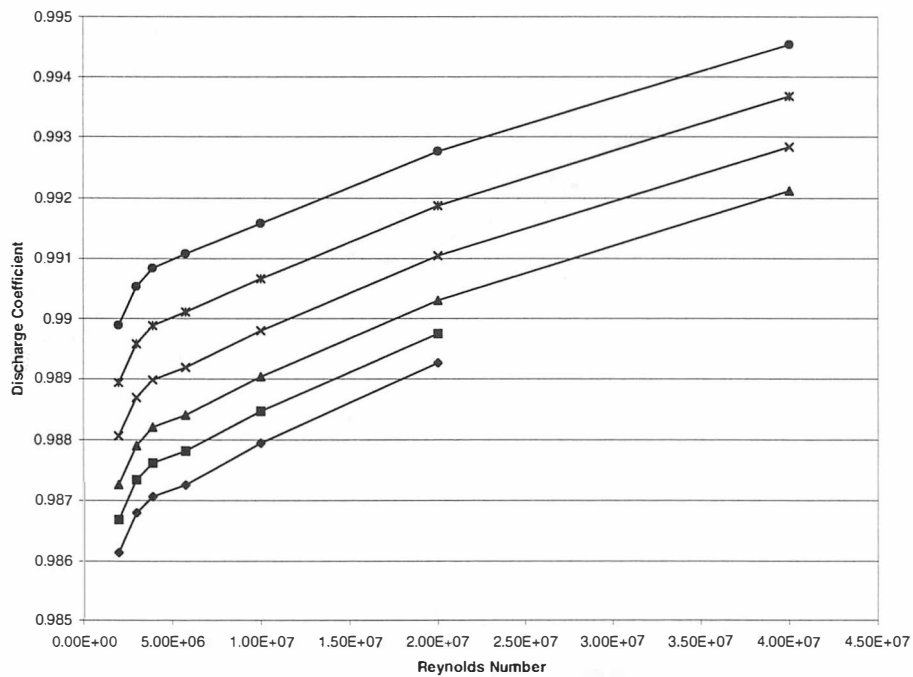


Figure 79. Discharge Coefficient Variation with Reynolds No. at Station 3.

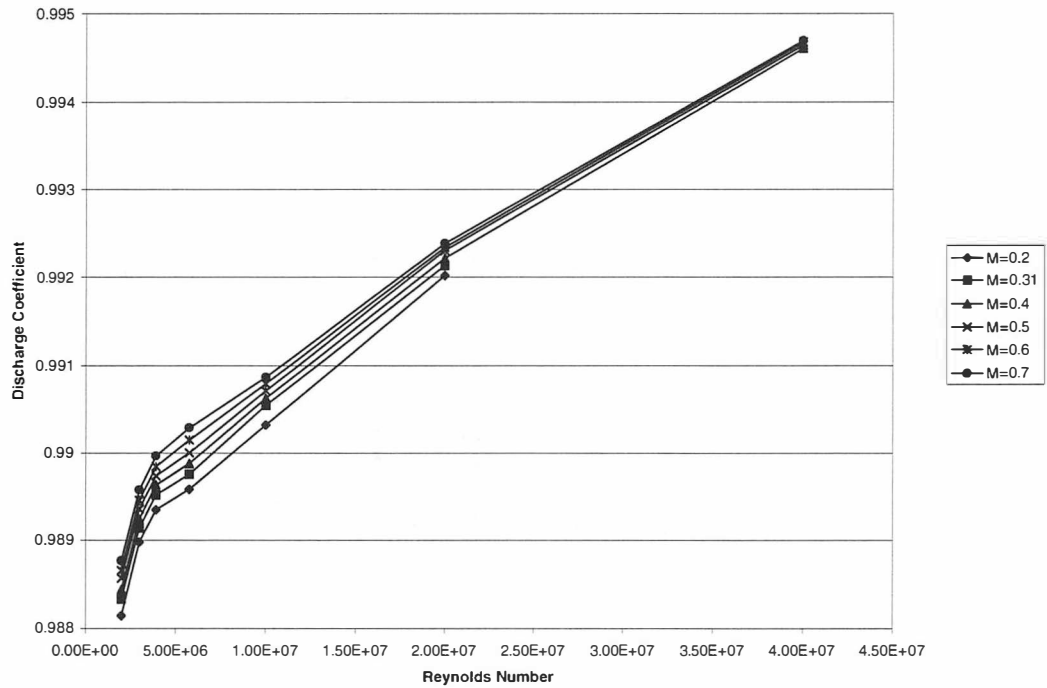


Figure 80. Discharge Coefficient Variation with Reynolds No. at Station 8.

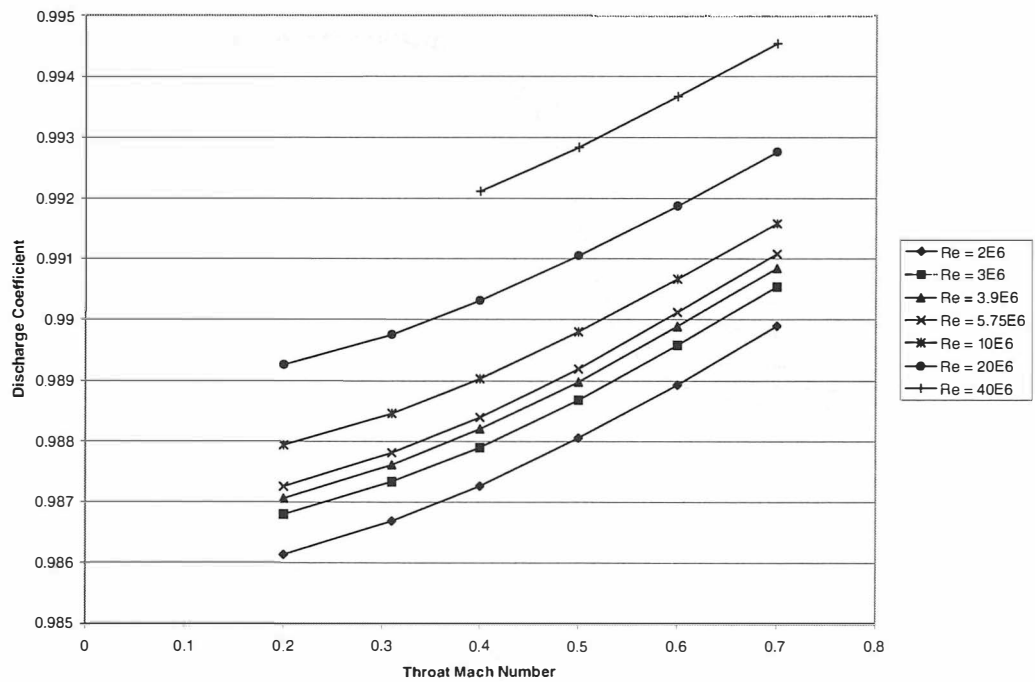


Figure 81. Discharge Coefficient Variation with Mach Number at Station 3.

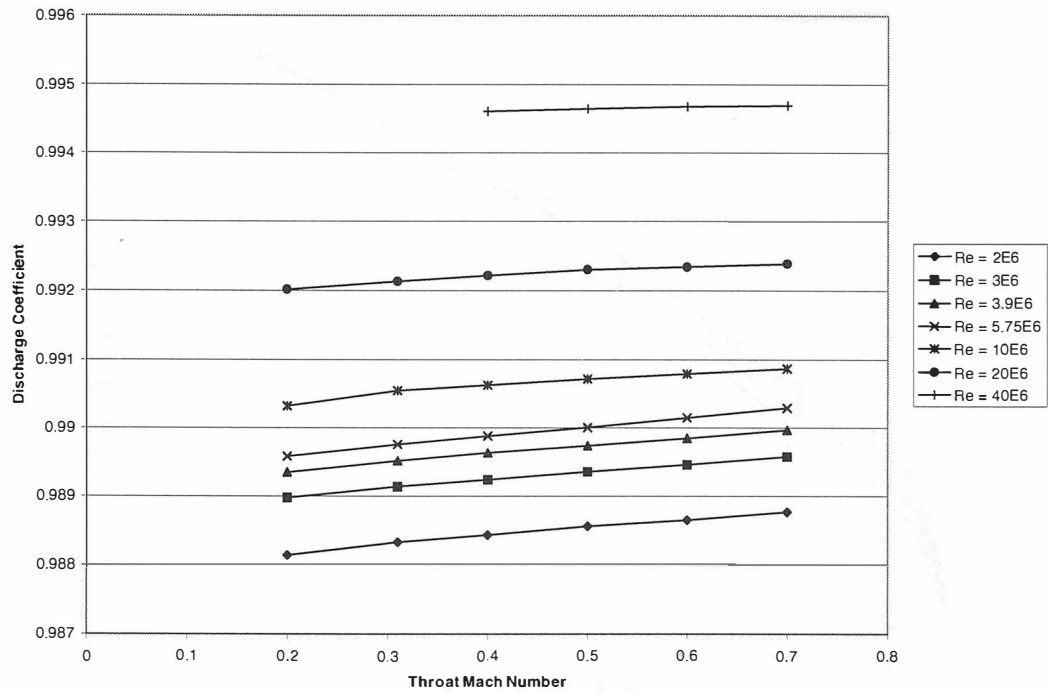


Figure 82. Discharge Coefficient Variation with Mach Number at Station 8.

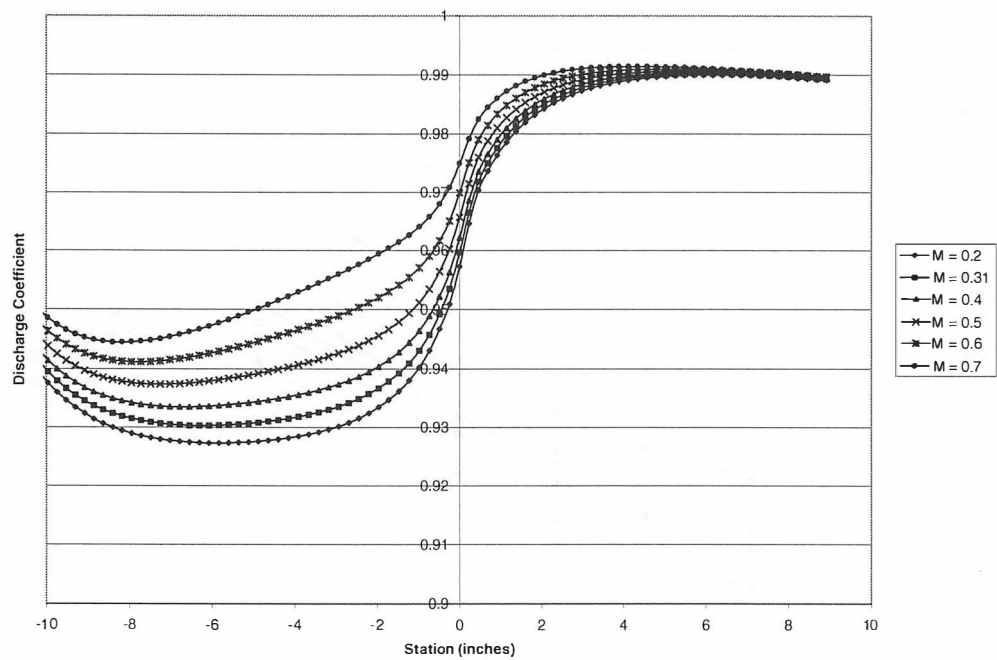


Figure 83. Axial Variation of Discharge Coefficient for Various Mach Numbers, Reynolds No. = 5.75E6.

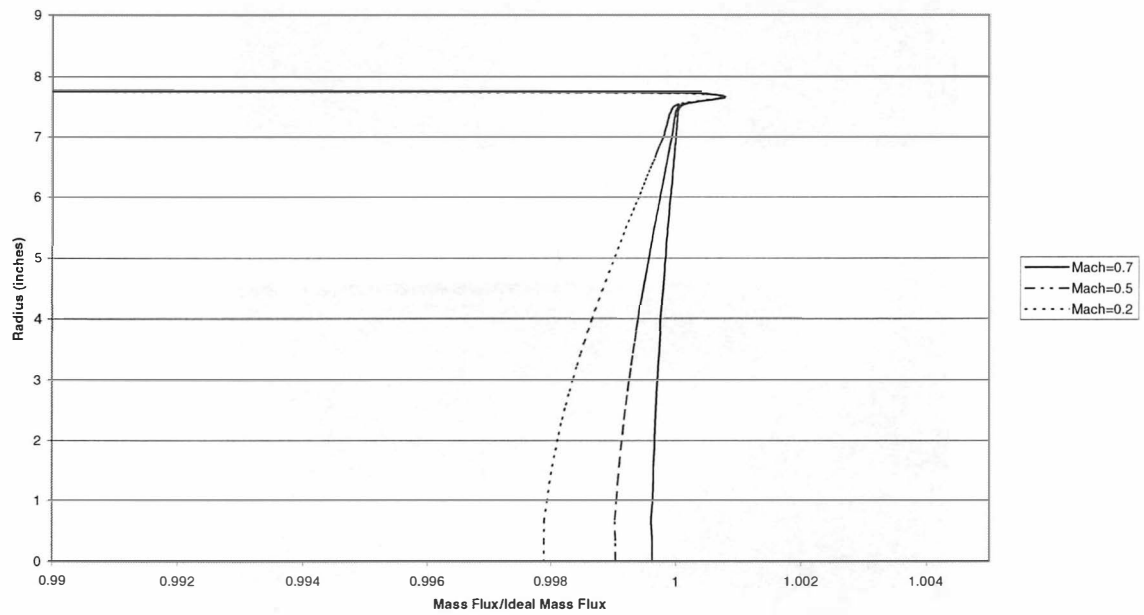
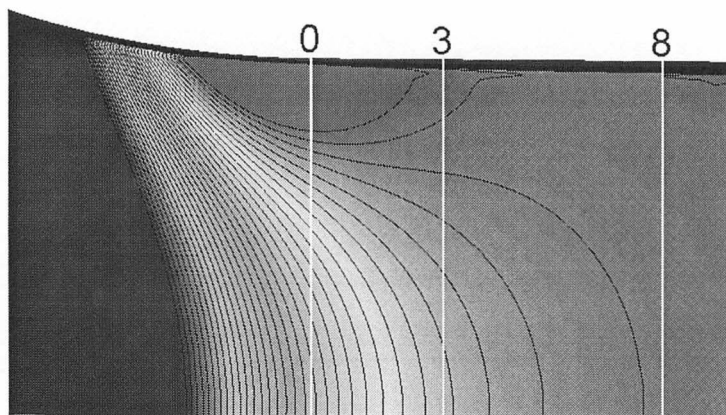
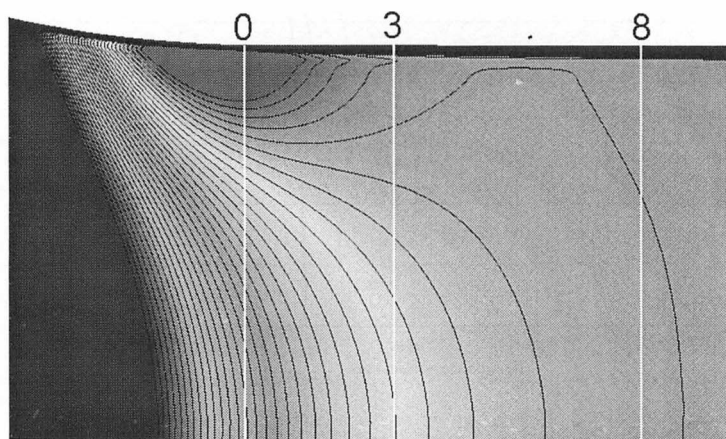


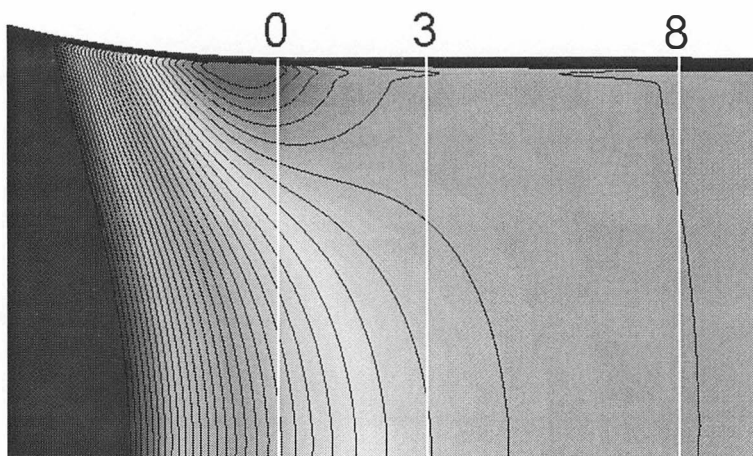
Figure 84. Comparison of Predicted Mass Flux at Station 8, Reynolds No. = 5.75E6.



$M = 0.2$



$M = 0.5$



$M = 0.5$

Figure 85. Comparison of Mass Flux Contours, $Re = 2E6$.

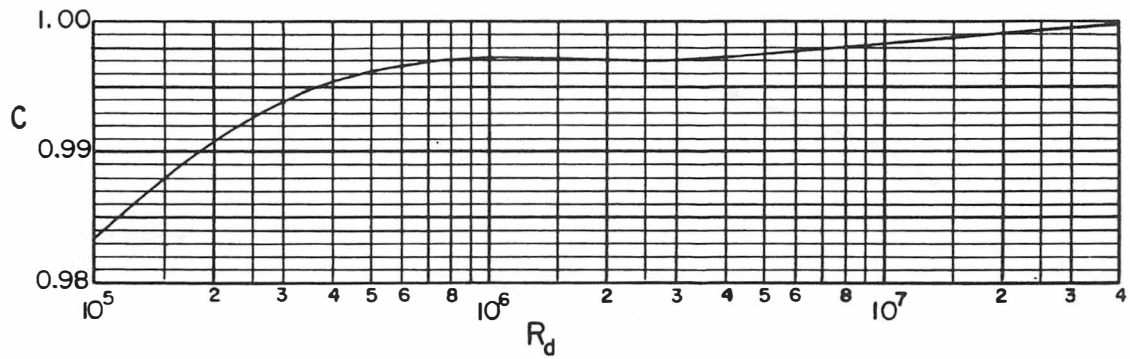


Figure 86. Calibration Curve of a Low Beta Flow Nozzle with Throat Pressure Taps.

Source: American Society of Mechanical Engineers Research Committee on Fluid Meters, Application, Part II of Fluid Meters, 6th ed, New York: The American Society of Mechanical Engineers, 1972.

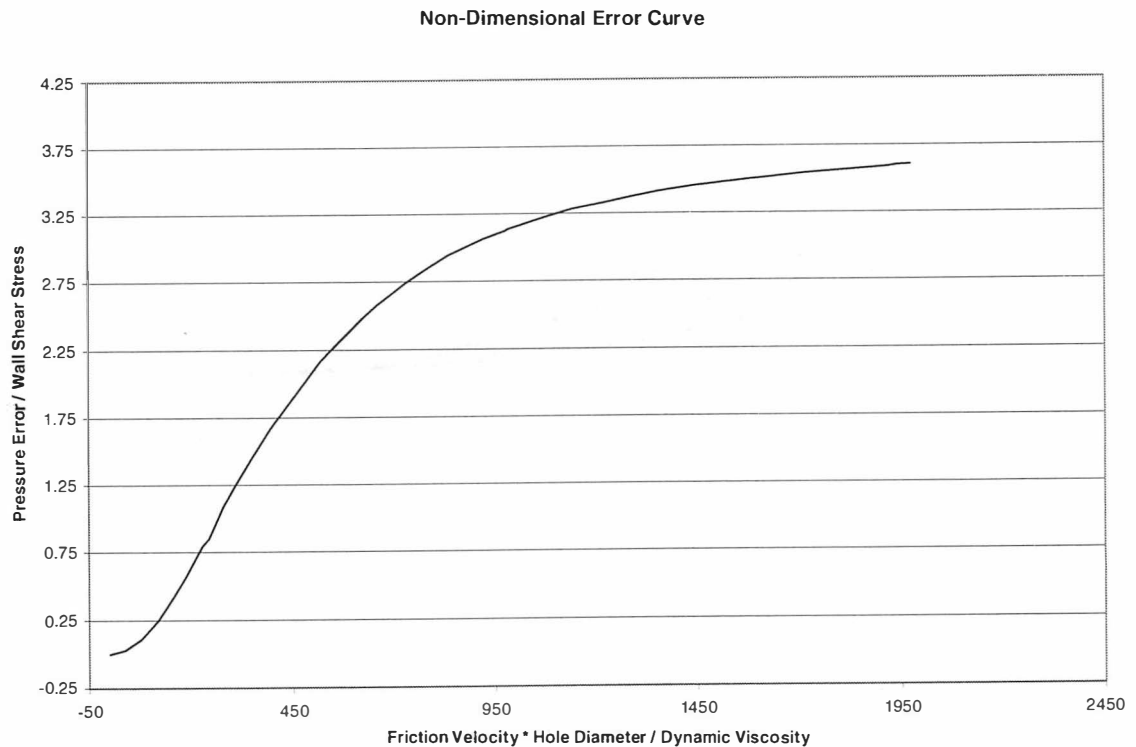


Figure 87. Static Pressure Tap Error as a Function of Wall Shear Stress and Hole Diameter.

Source: Franklin, R. E. and Wallace, J.M., "Absolute Measurement of Static-Hole Error Using Flush Transducers," *Journal of Fluid Mechanics*, Vol. 42, Part 1, 1070, pp. 33-48.

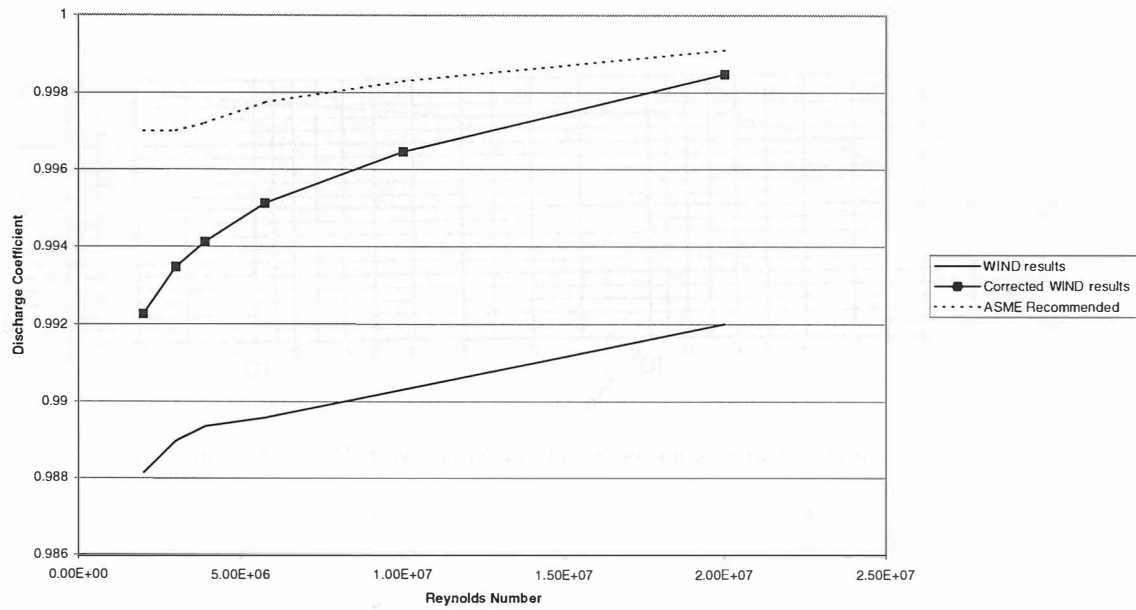


Figure 88. Comparison of Computed Results at Mach 0.2 with ASME Recommended Values.

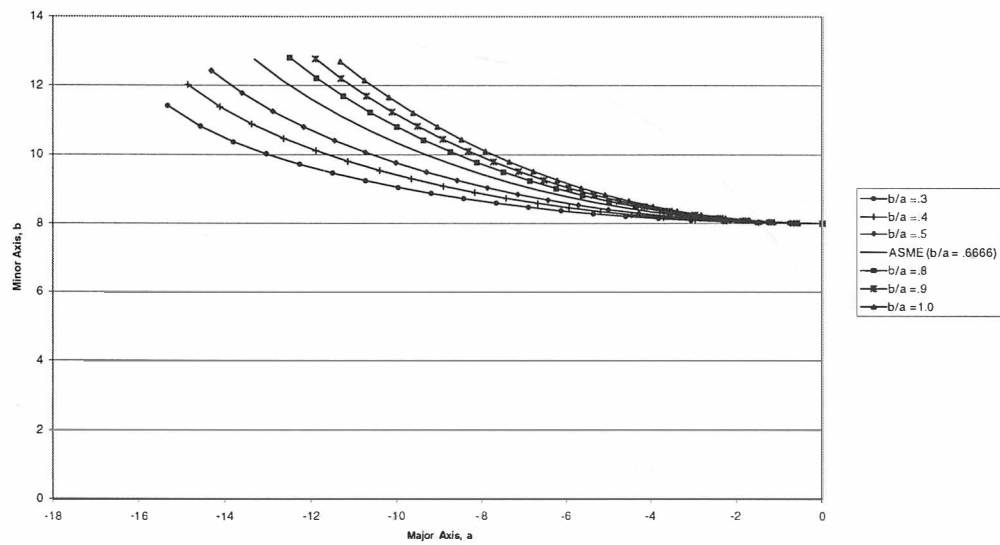


Figure 89. Range of Elliptical Inlet Profiles for Bellmouth (a – major axis, b – minor axis).

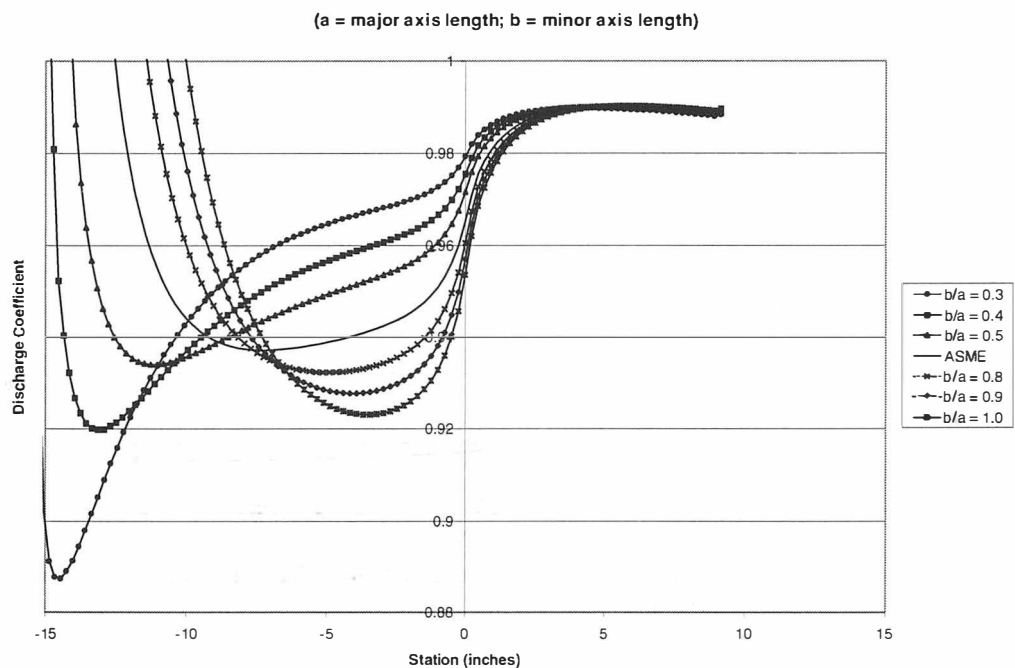


Figure 90. Axial Distribution of the Computed Discharge Coefficient for Various Axis Ratios, $M=0.5$, $Re=3E6$.

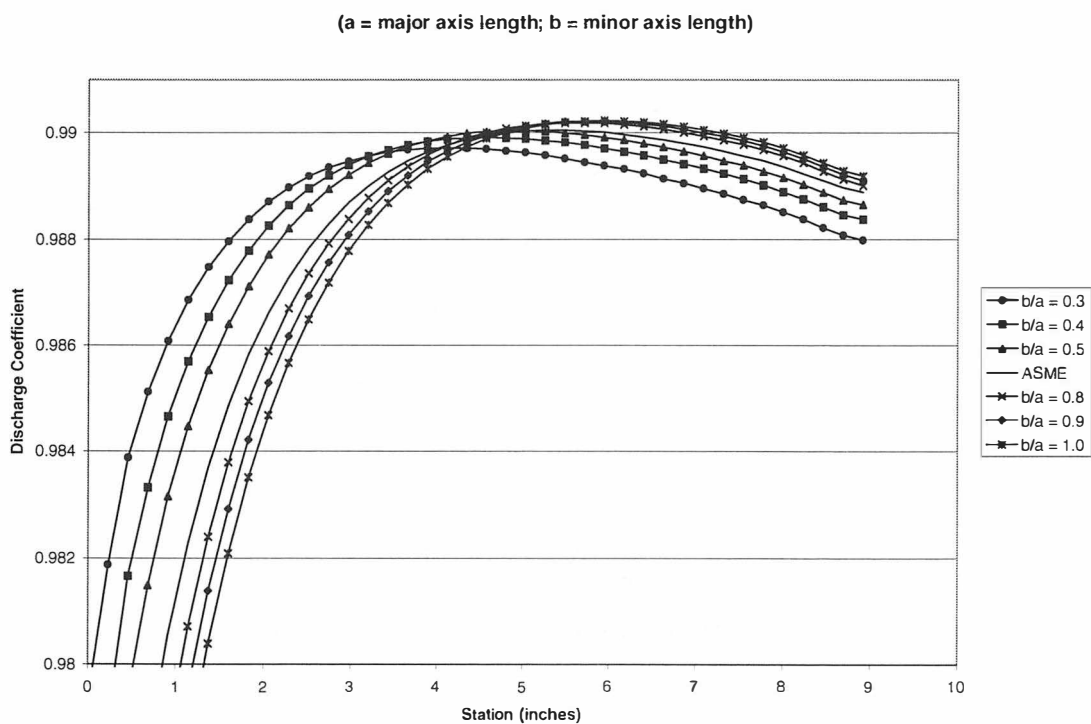


Figure 91. Axial Distribution of the Computed Discharge Coefficient for Various Axis Ratios in the Throat Region, $M = 0.5$, $Re = 3E6$.

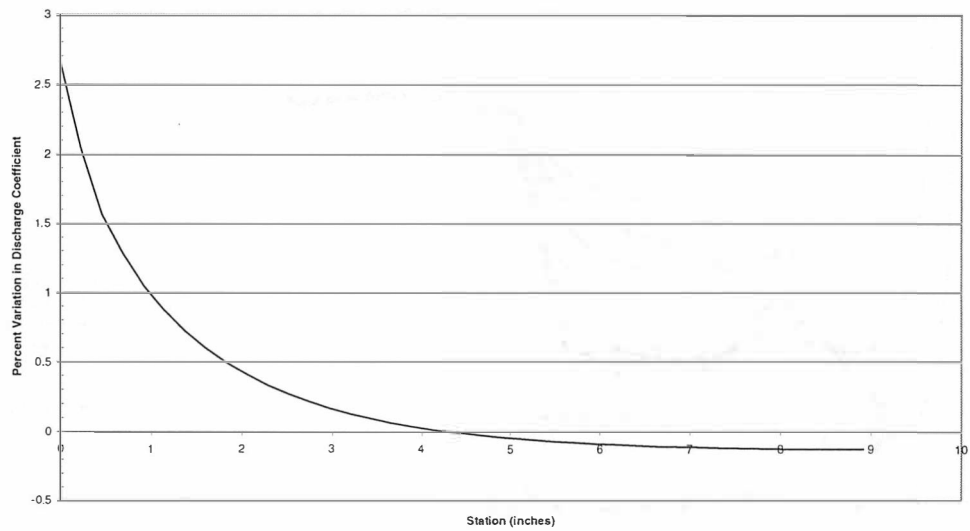


Figure 92. Axial Distribution of the Percent Variation of the Computed Discharge Coefficient Between $b/a = 0.3$ and $b/a = 1.0$; $M=0.5$, $Re = 3E6$.

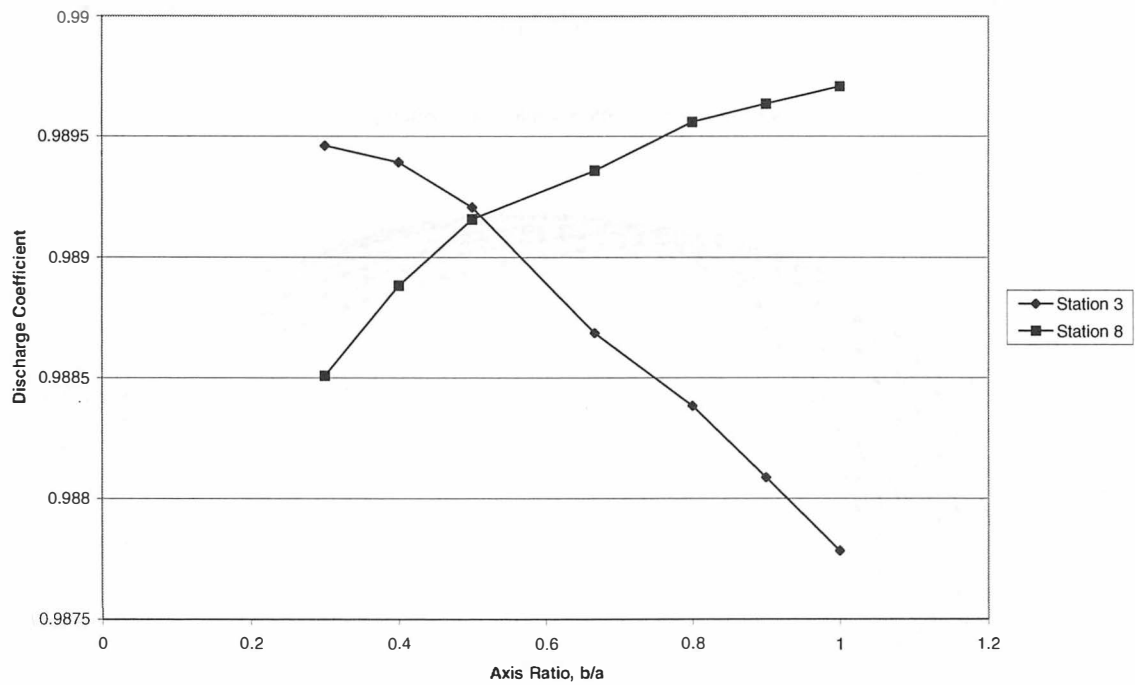


Figure 93. Variation of the Computed Discharge Coefficient as a Function of Axis Ratio, $M=0.5$, $Re=3E6$.

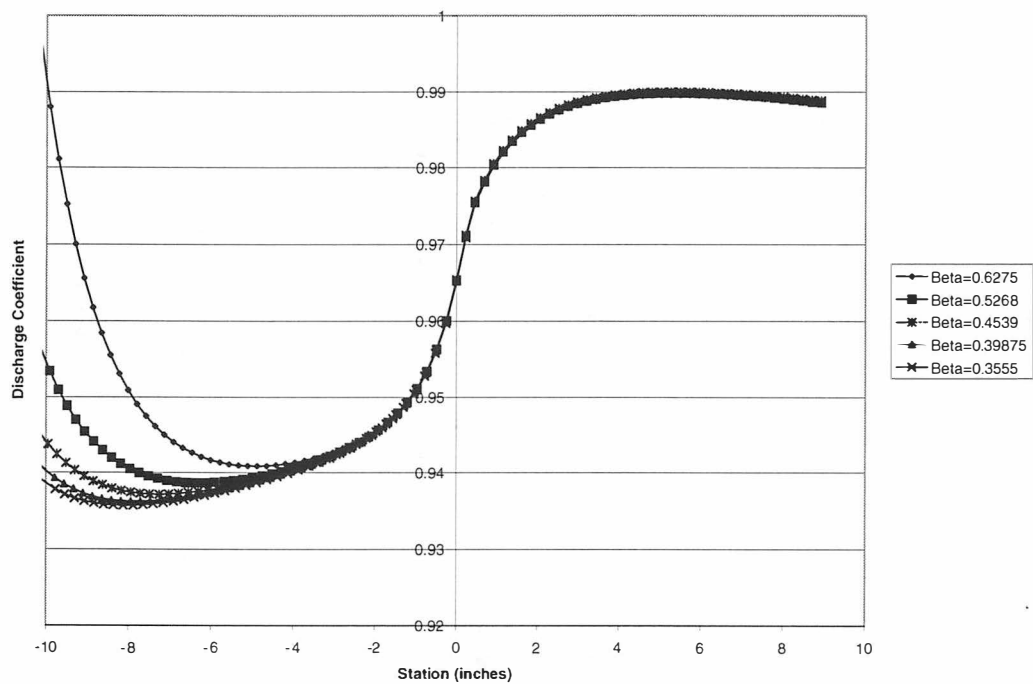


Figure 94. Axial Distribution of the Computed Discharge Coefficient for Various Beta Ratios, $M = 0.5$, $Re = 3E6$.

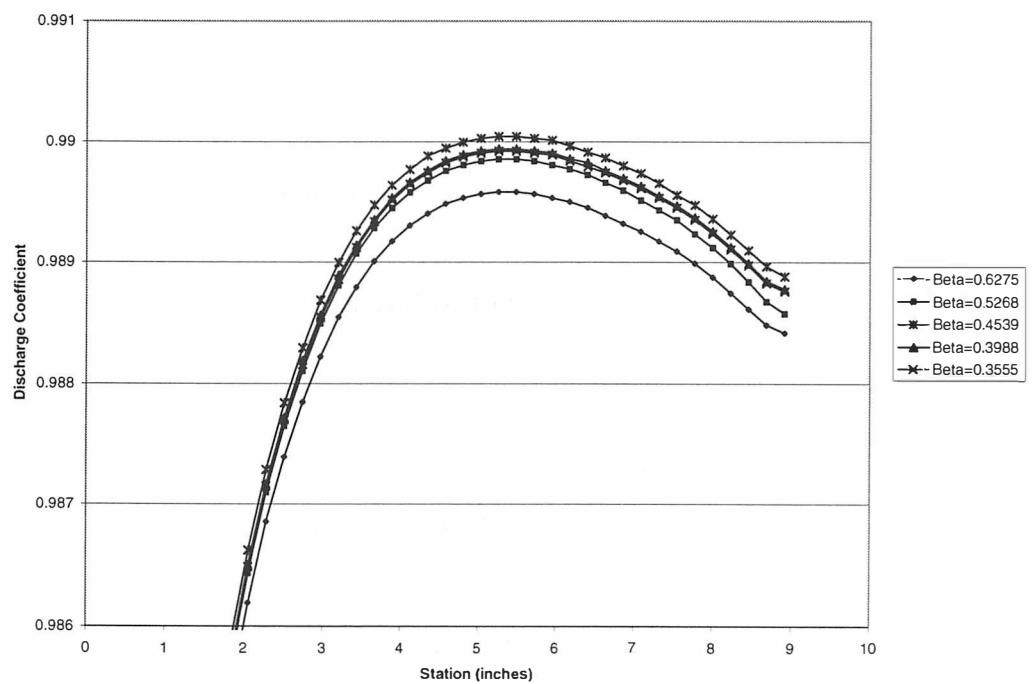


Figure 95. Axial Distribution of the Computed Discharge Coefficient for Various Beta Ratios in Throat Region, $M = 0.5$, $Re = 3E6$.

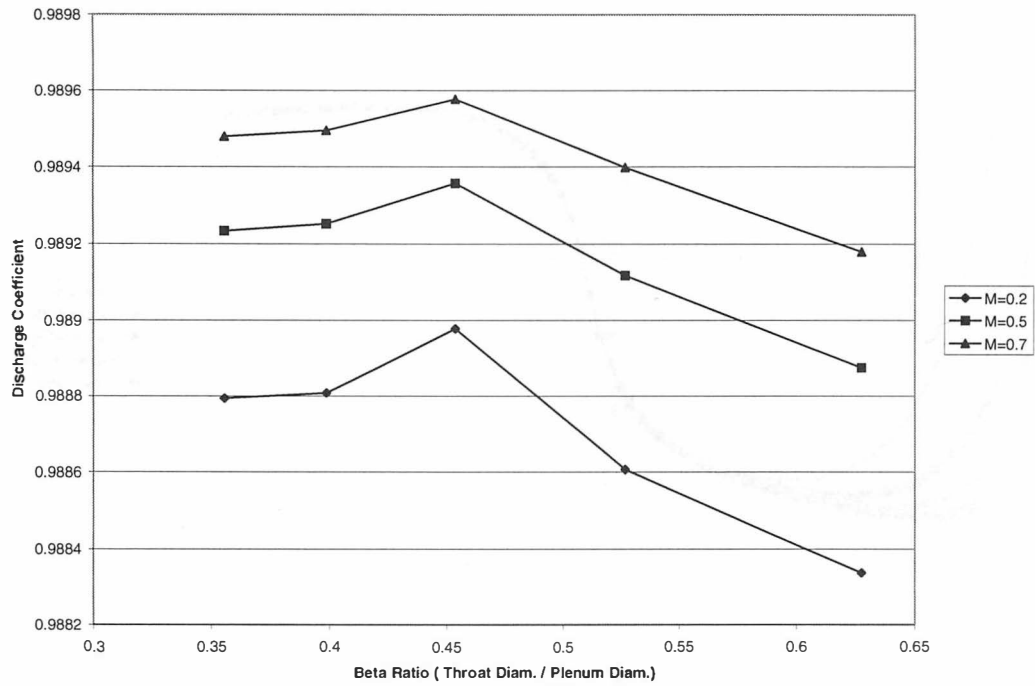
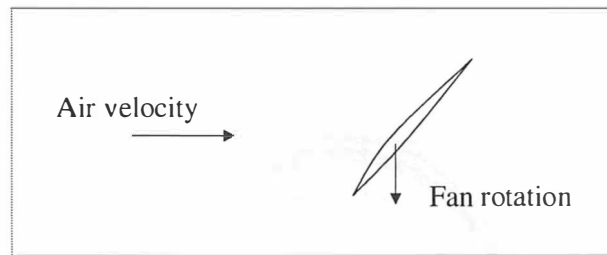
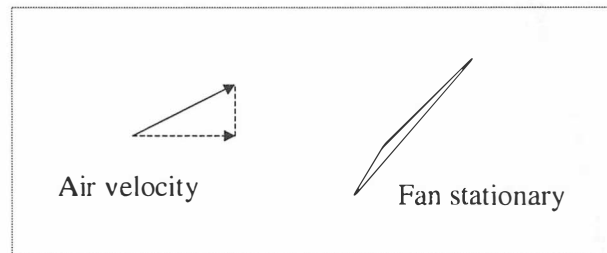


Figure 96. Variation of Discharge Coefficient with Beta Ratio.



Inertial Frame of Reference



Rotating Frame of Reference

Figure 97. Comparison of Reference Frames.

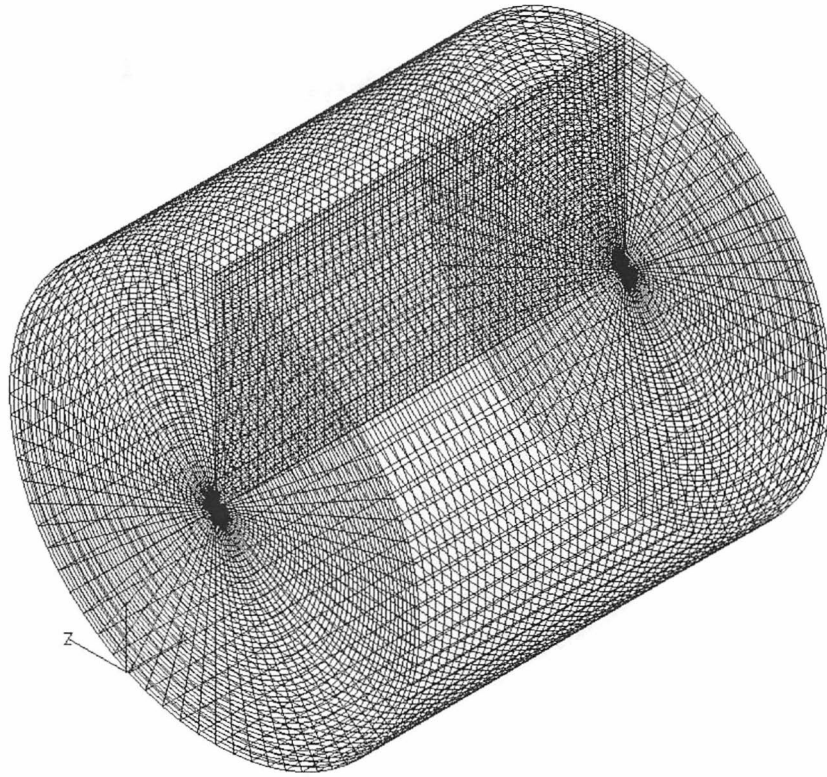


Figure 98. Cylinder Test Case Grid.

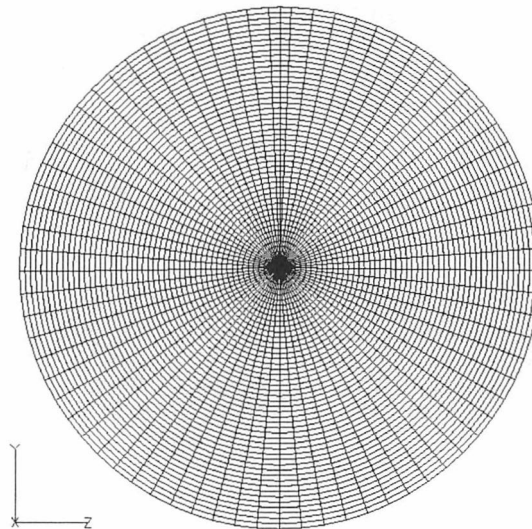


Figure 99. Cylinder Test Case Grid: End View.

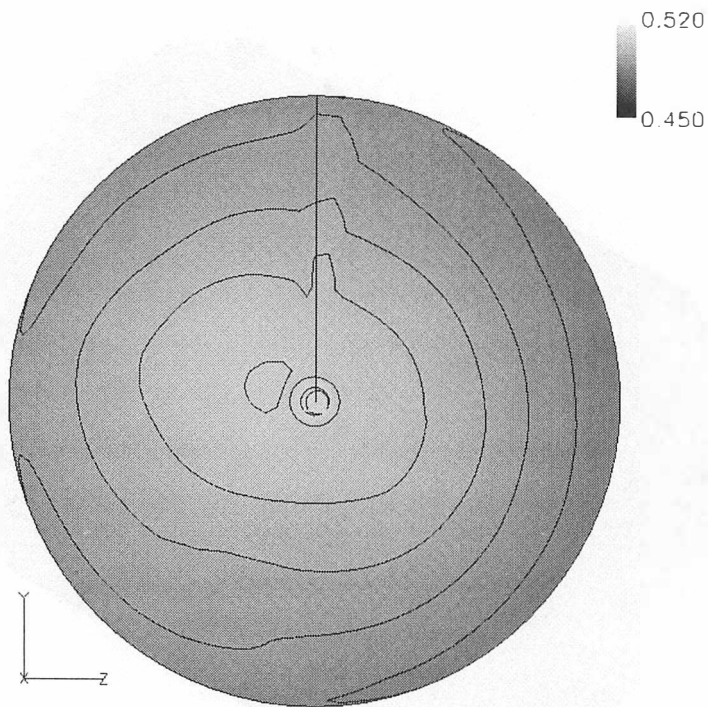


Figure 100. Mach Contours for Rotating Frame Test Case.



Figure 101. Mach Contours for Inertial Frame Test Case.

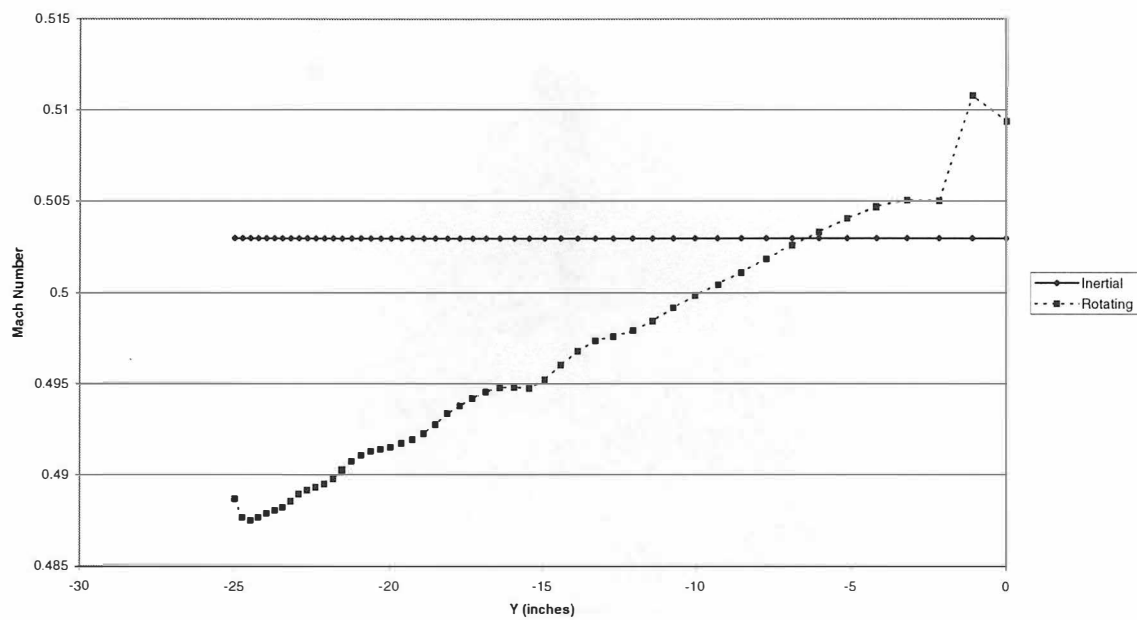


Figure 102. Comparison of Predicted Mach Number Along Vertical Radius.

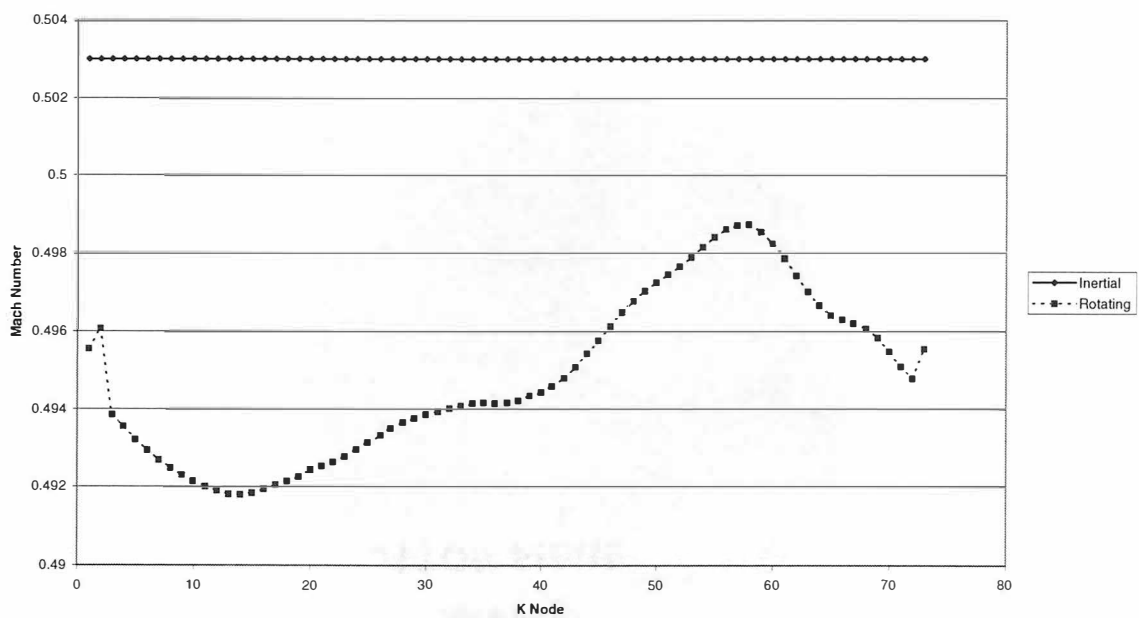


Figure 103. Comparison of Predicted Mach Number Along Circumferential $j=25$ Grid Line.

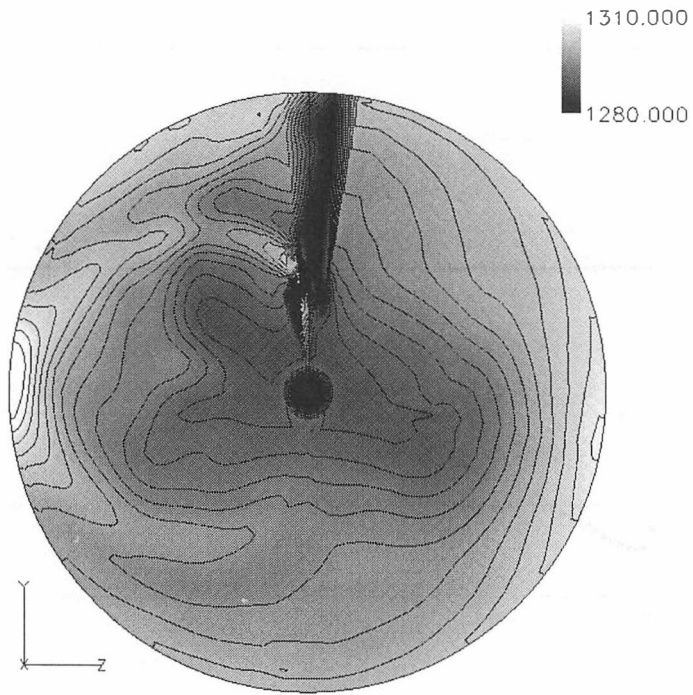


Figure 104. Predicted Pressure Contours for Rotating Frame Test Case.

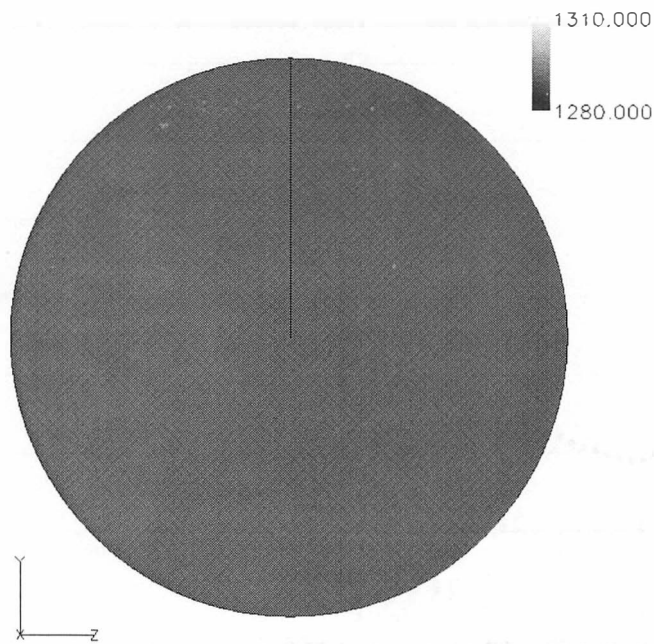


Figure 105. Predicted Pressure Contours for Inertial Frame Test Case.

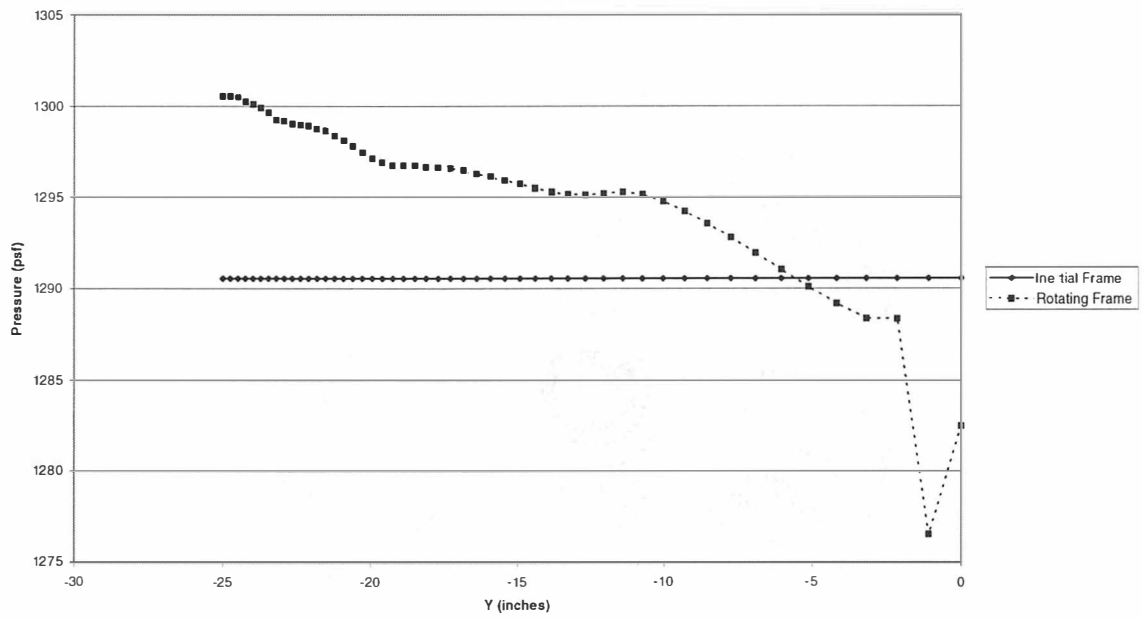


Figure 106. Comparison of Predicted Pressure Along Vertical Radius.

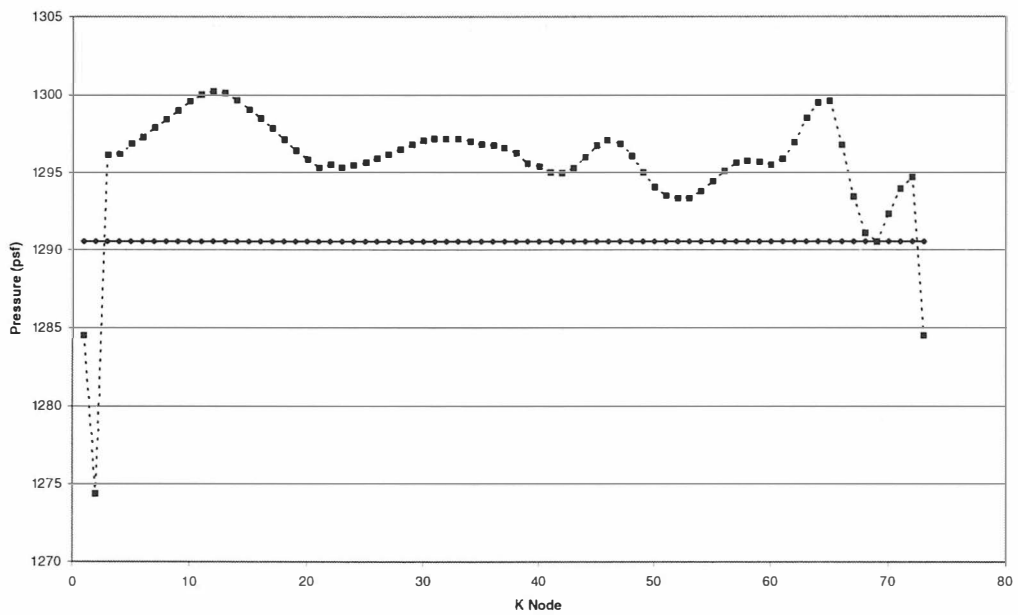


Figure 107. Comparison of Predicted Pressure Along j=25 Grid Line.

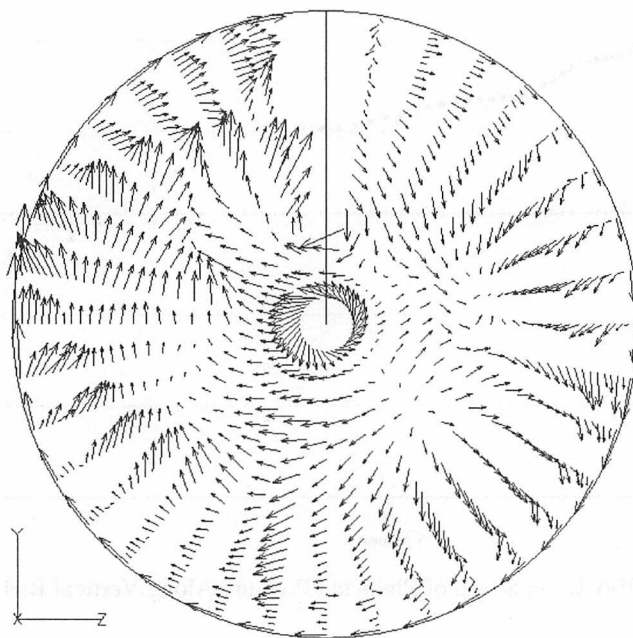


Figure 108. Flow Vectors in the Y-Z Plane for Rotating Frame Test Case.

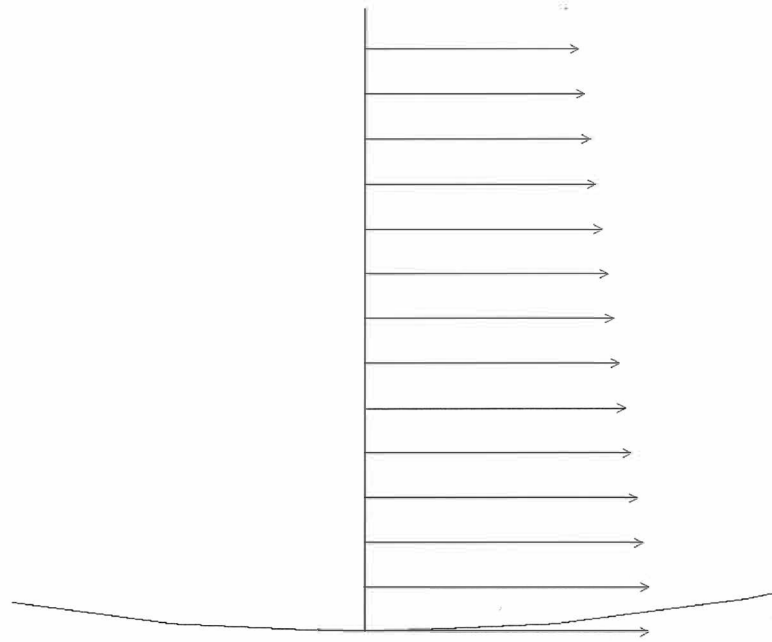


Figure 109. Velocity Vectors in Rotating Frame At Uncorrected Slip Wall Boundary.

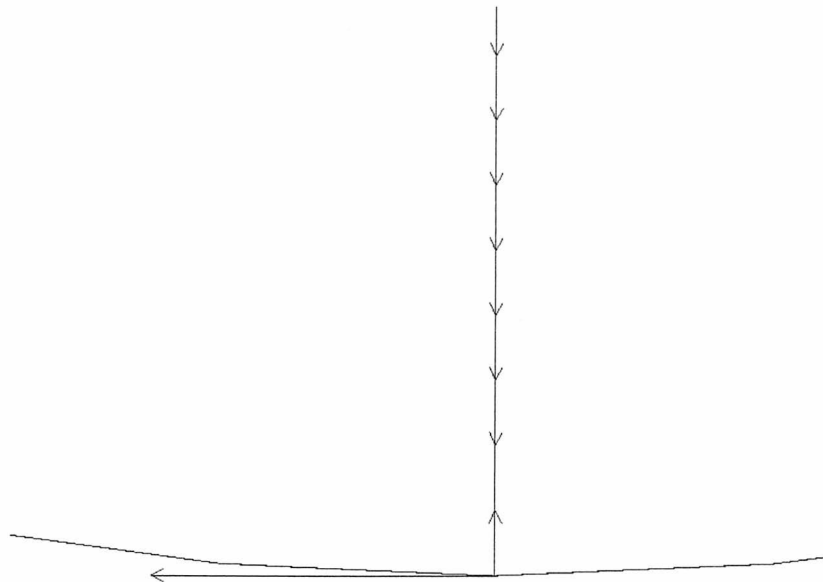


Figure 110. Velocity Vectors in Inertial Frame At Uncorrected Slip Wall Boundary.

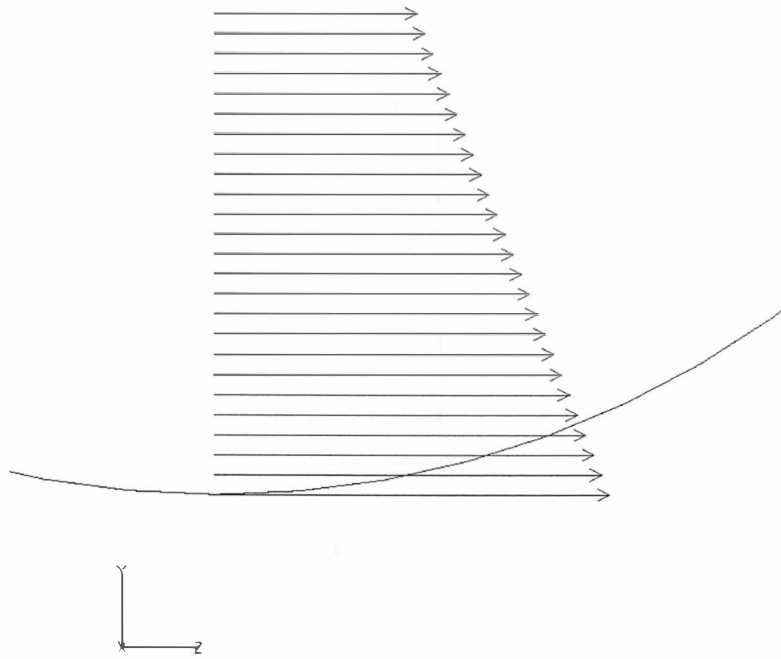


Figure 111. Velocity Vectors in Rotating Frame At Corrected Slip Wall Boundary.

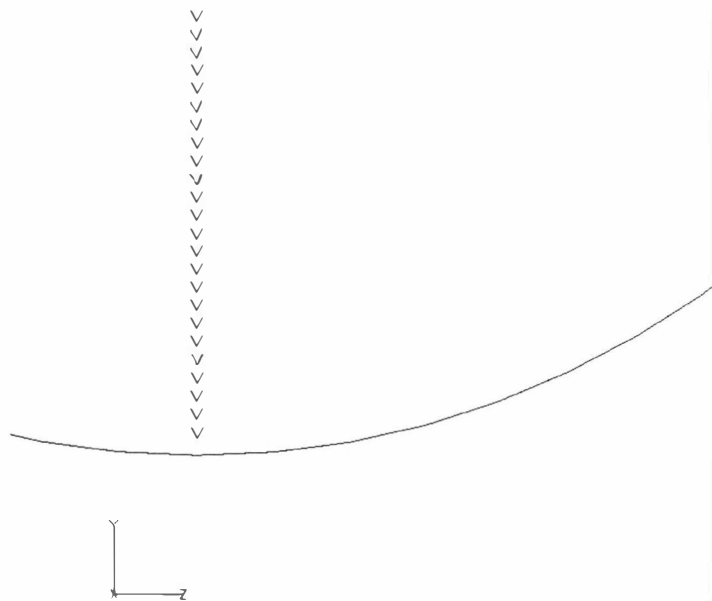


Figure 112. Velocity Vectors in Inertial Frame at Corrected Slip Wall Boundary.

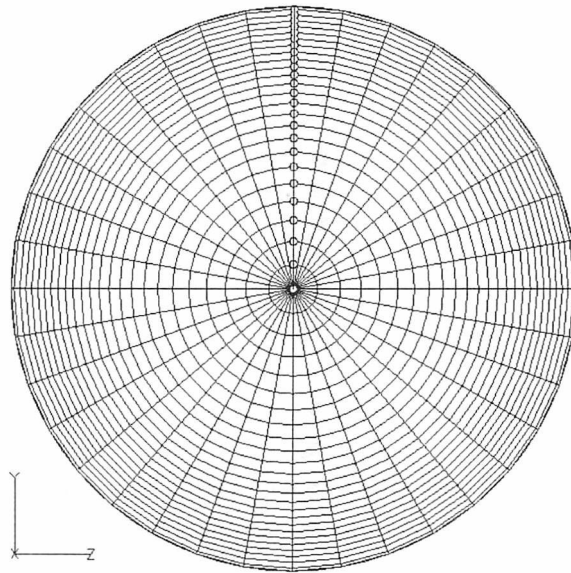


Figure 113. Cylindrical Test Grid With Coupled Boundary Highlighted.

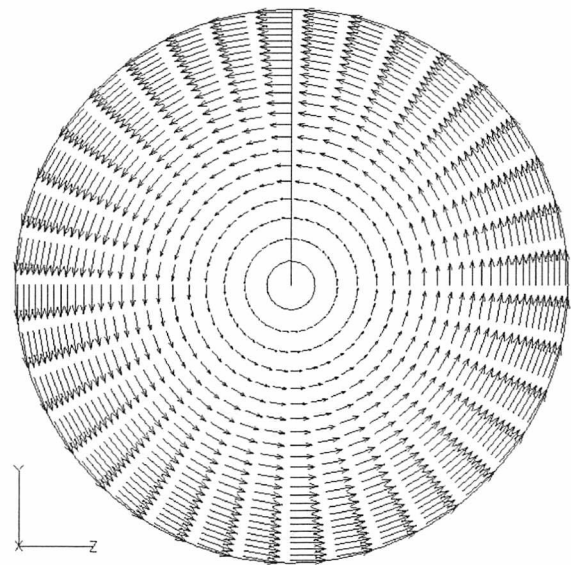


Figure 114. Velocity Vectors in Rotating Frame for Cylindrical Test Case.

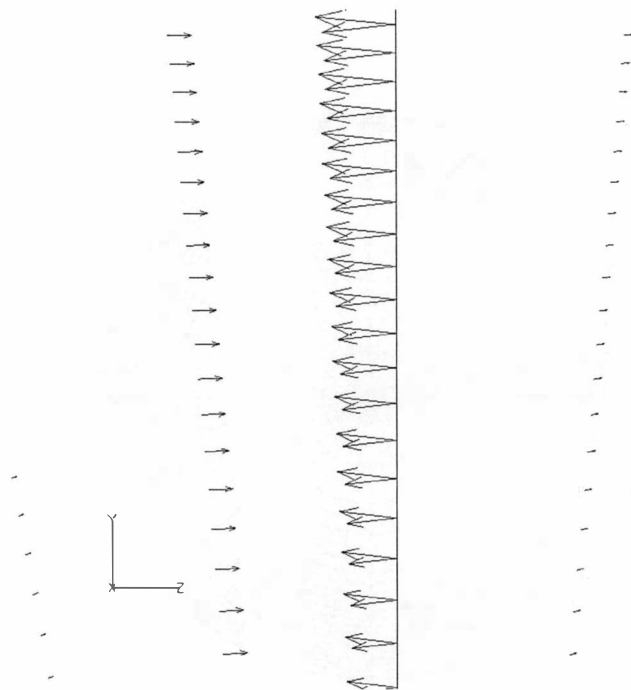


Figure 115. Velocity Vectors in Inertial Frame at Uncorrected Coupled Boundary.

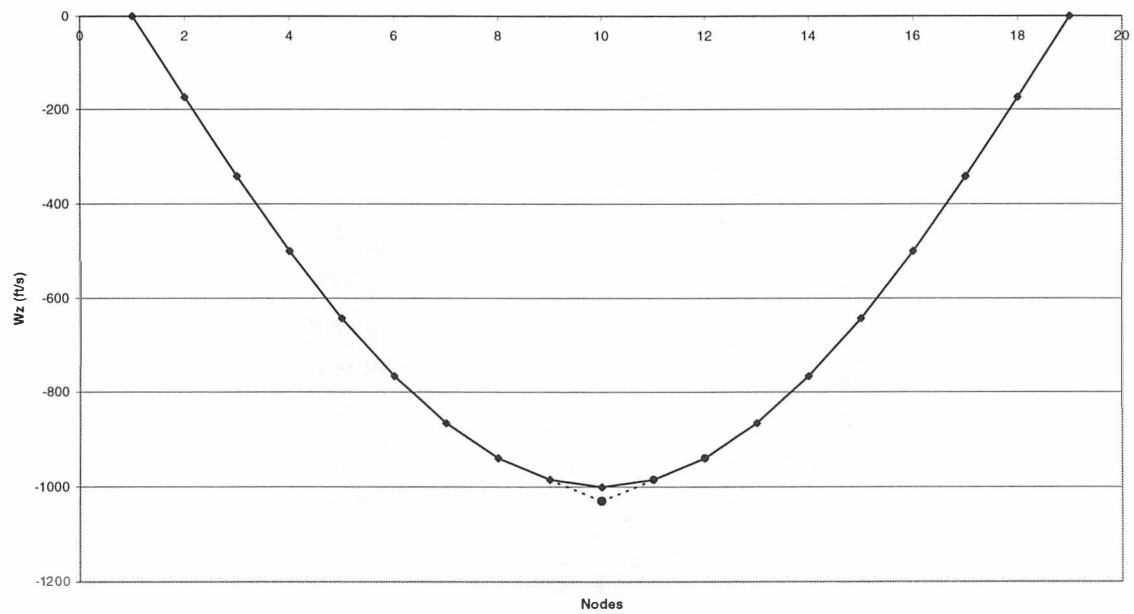


Figure 116. Illustration of Coupling Error for Z Velocity Component on Vertical Coupled Boundary.

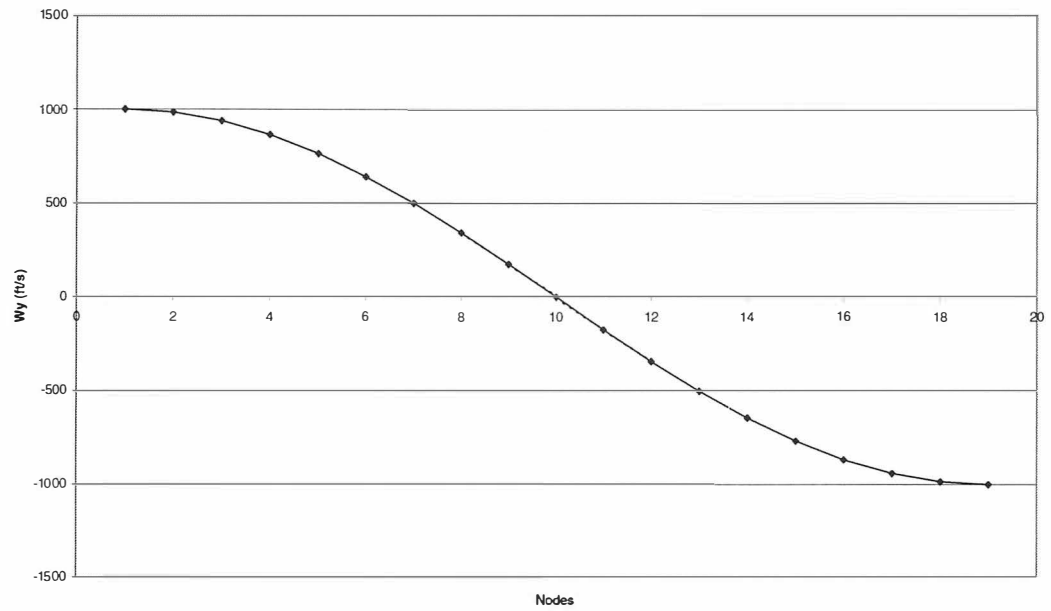


Figure 117. Illustration of Coupling Error for Y Velocity Component on Vertical Coupled Boundary.

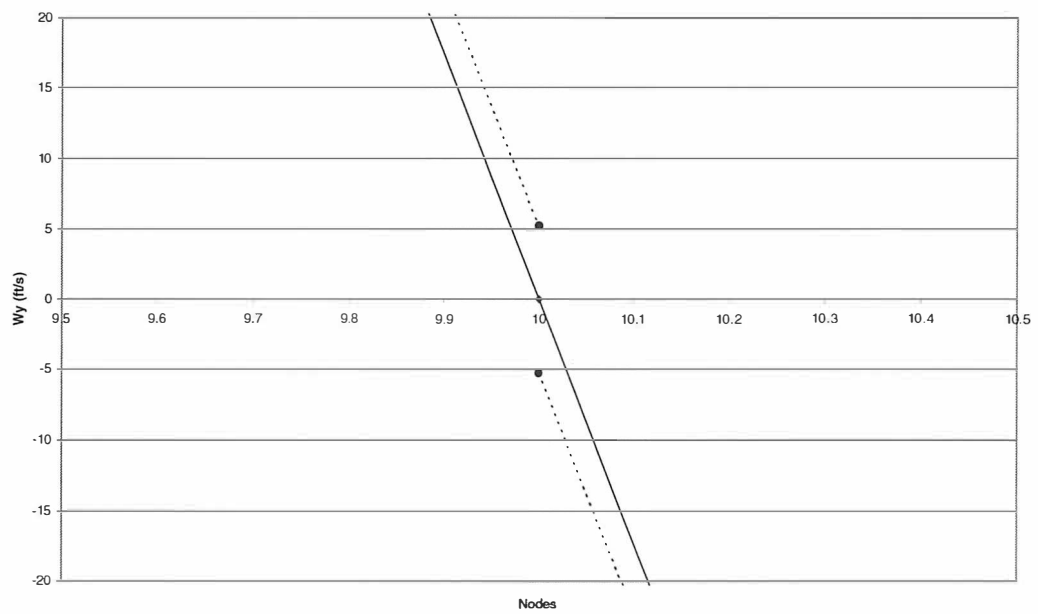


Figure 118. Detail of Coupling Error for Y Velocity Component on Vertical Coupled Boundary.

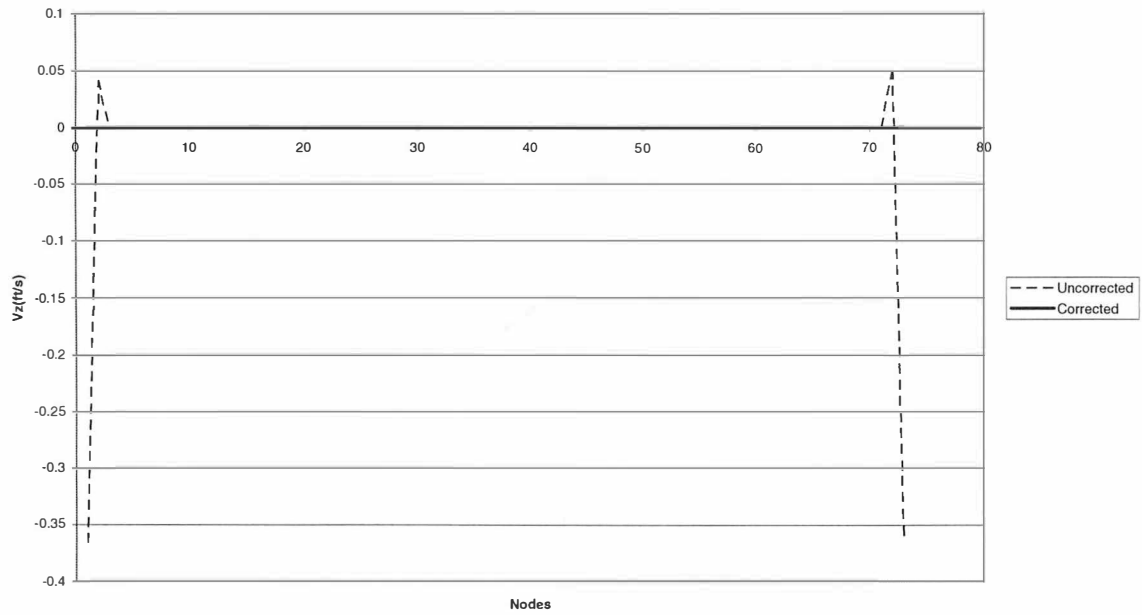


Figure 119. Comparison of Z Component of Velocity for Corrected and Uncorrected Boundary Layer Coupling.

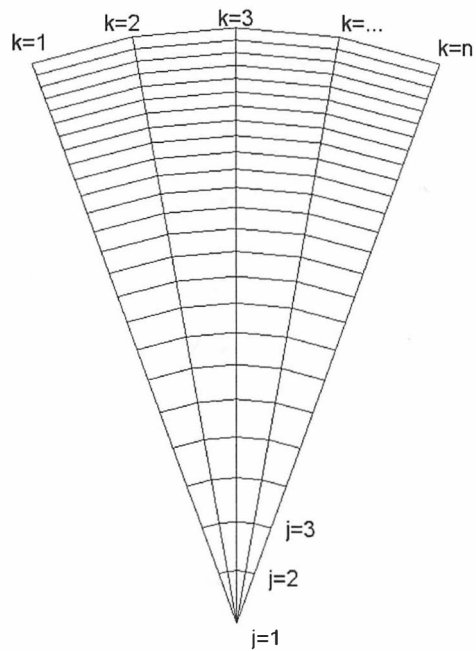


Figure 120: Singular Axis Averaging and Relaxation Schematic.

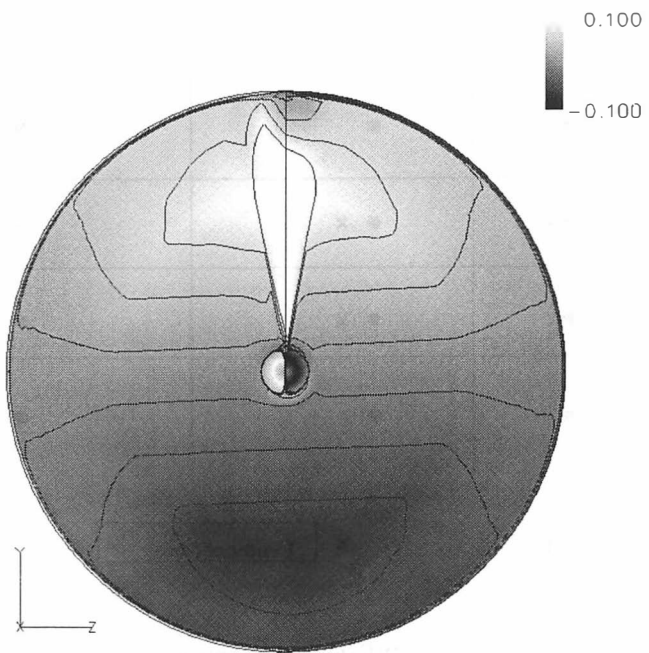


Figure 121. Y Velocity Component After Single Iteration.

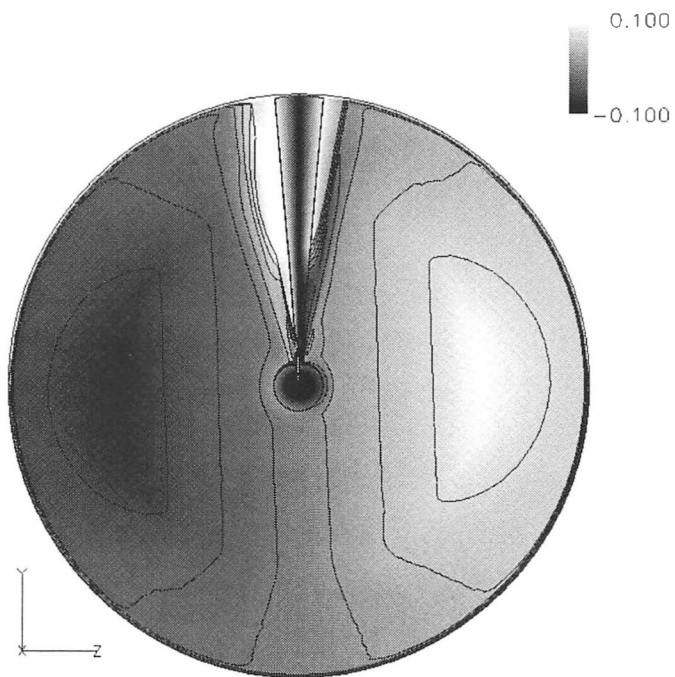


Figure 122. Z Velocity Component After Single Iteration.

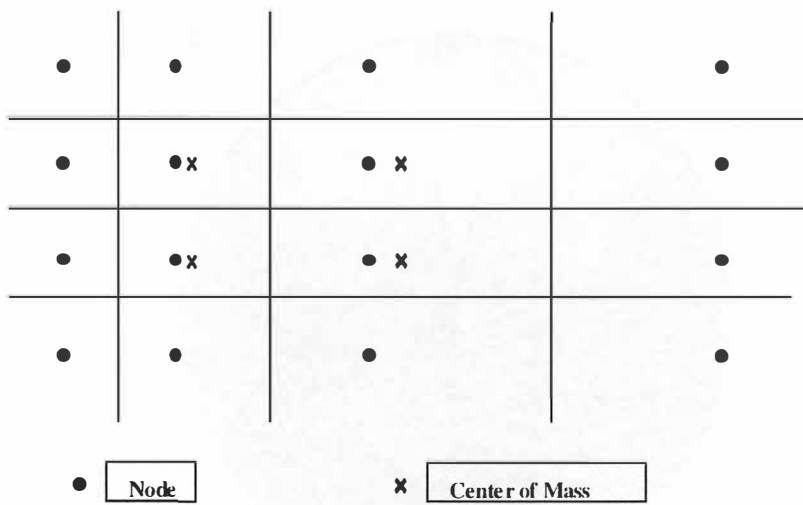
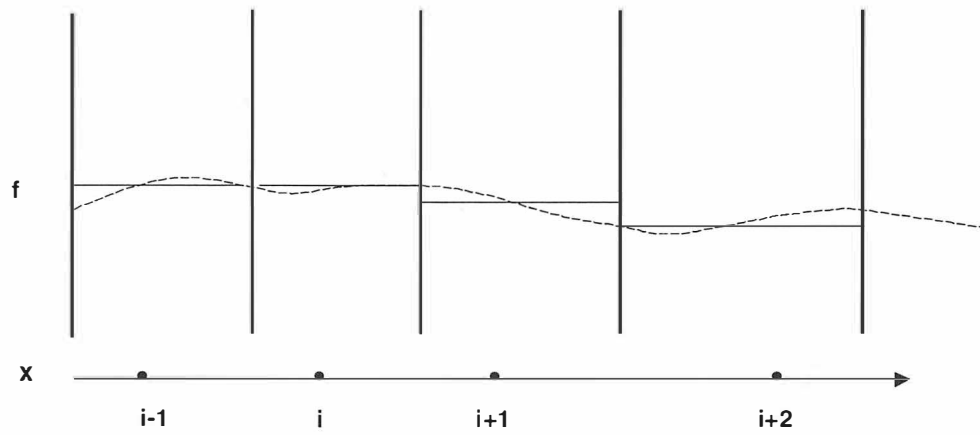
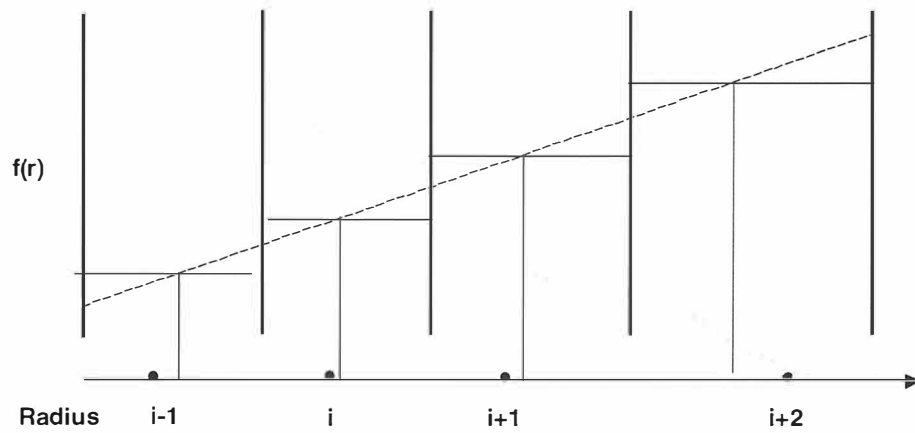


Figure 123. WIND Code Control Volumes.



a. Roe Scheme in Inertial Frame For Arbitrary Function



b. Roe Scheme in Rotating Frame For $f = \text{Function of Radius}$

Figure 124. Roe Method Comparison of Inertial and Rotating Frame.

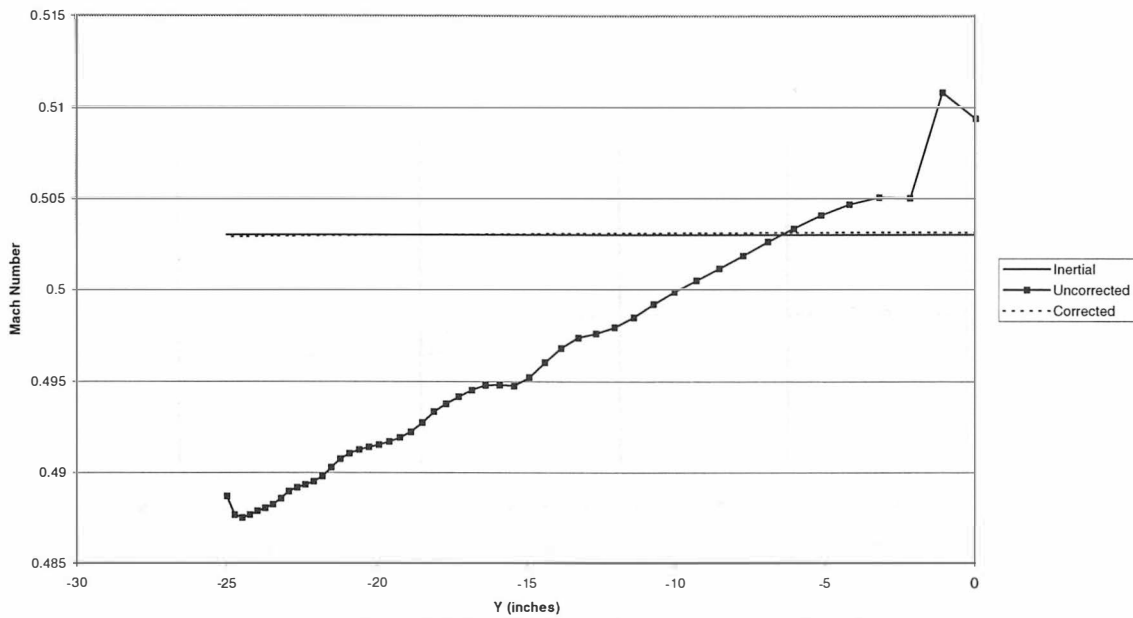


Figure 125. Comparison of Predicted Mach Number Along Vertical Radius of Simple Cylinder for Corrected and Uncorrected Rotation.

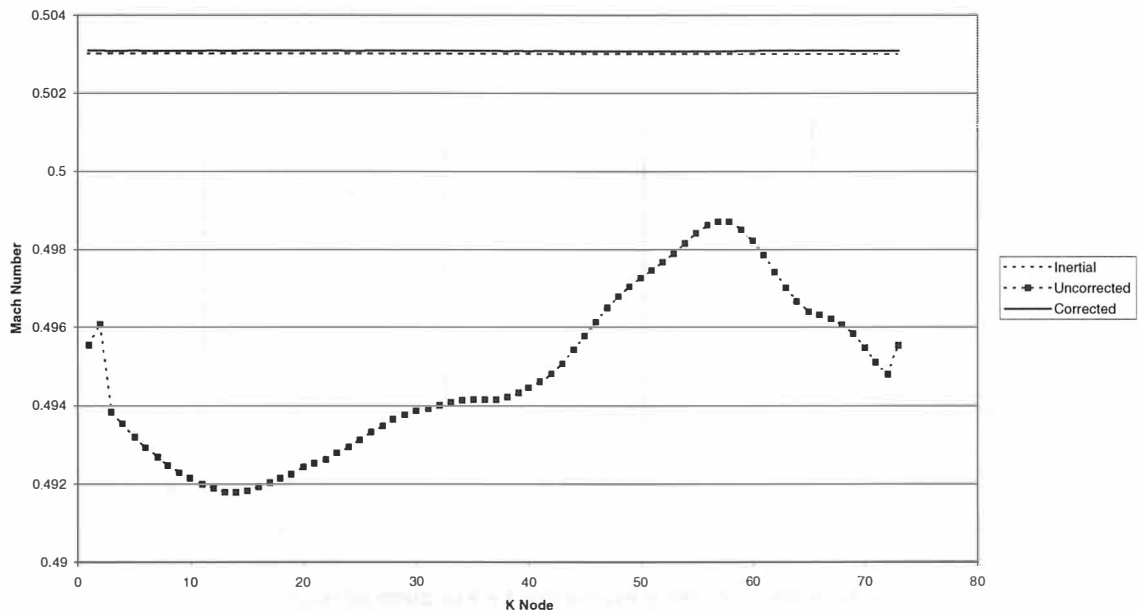


Figure 126. Comparison of Predicted Mach Number Along $j=25$ Grid Line for Corrected and Uncorrected Rotation.

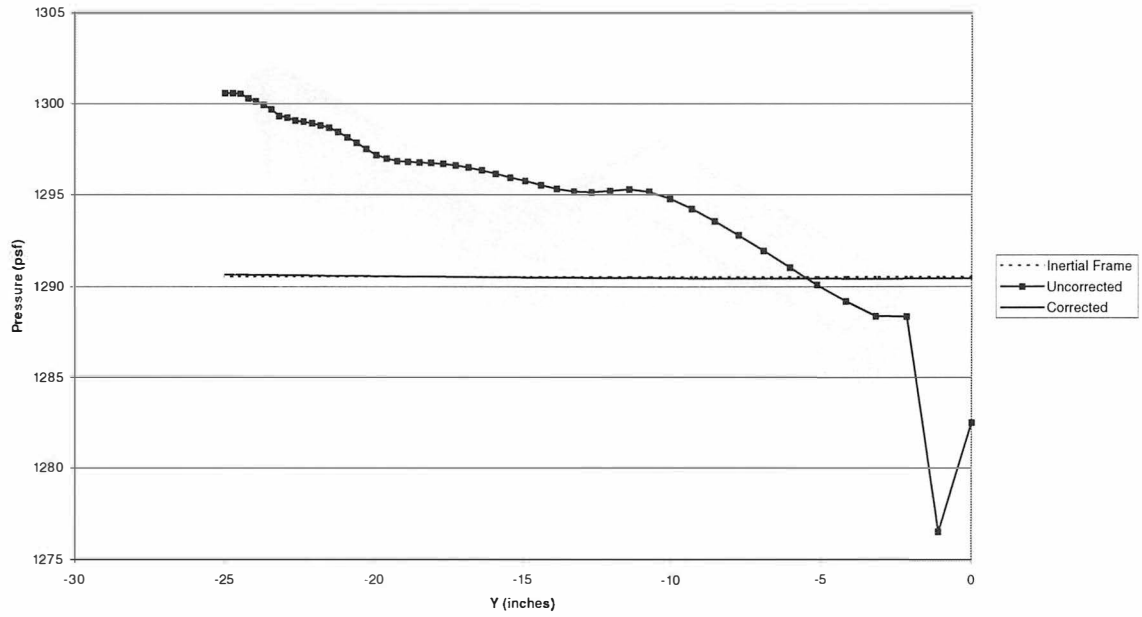


Figure 127. Comparison of Predicted Pressure Along Vertical Radius of Simple Cylinder for Corrected and Uncorrected Rotation.

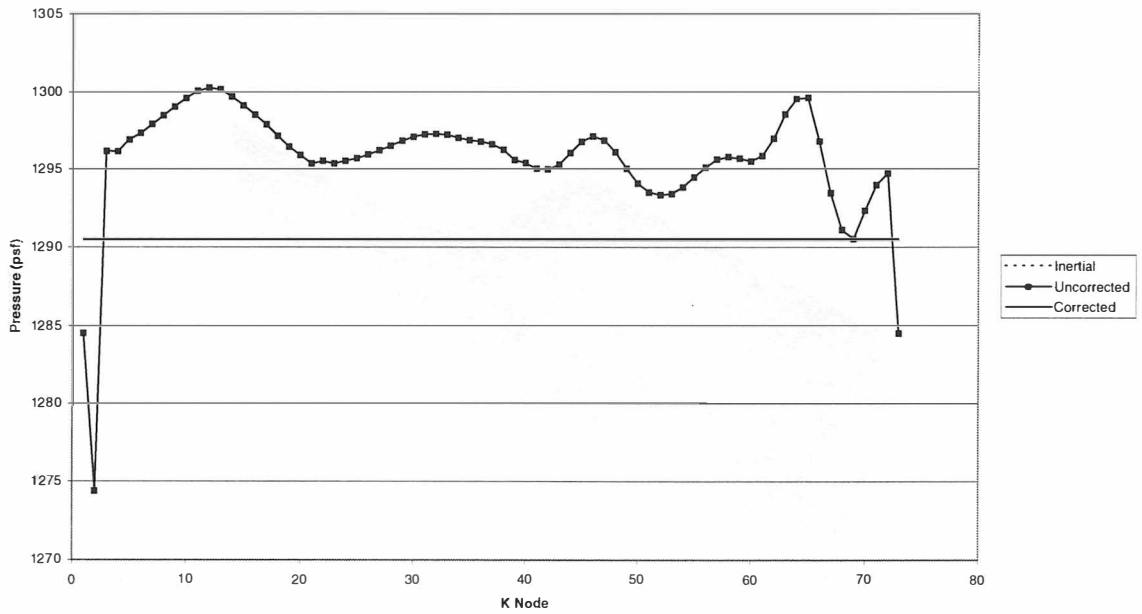


Figure 128. Comparison of Predicted Pressure Along Vertical Radius of Simple Cylinder for Corrected and Uncorrected Rotation.

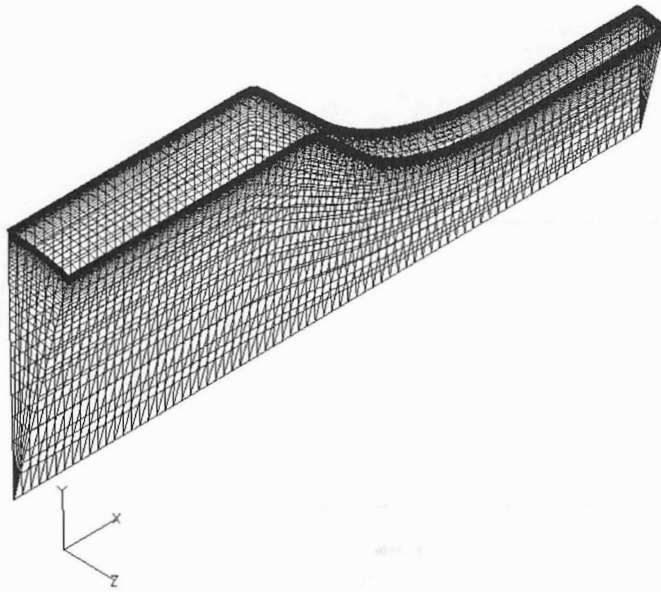


Figure 129. Three Dimensional Bellmouth Grid.

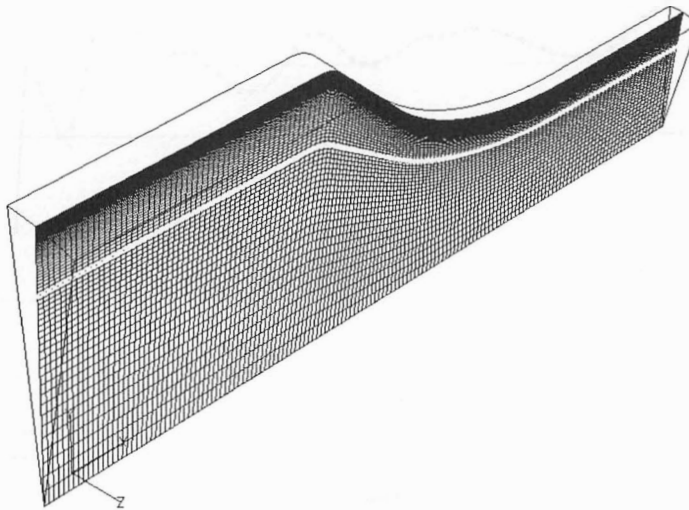


Figure 130. Grid in Single k-Plane for Comparison Results.

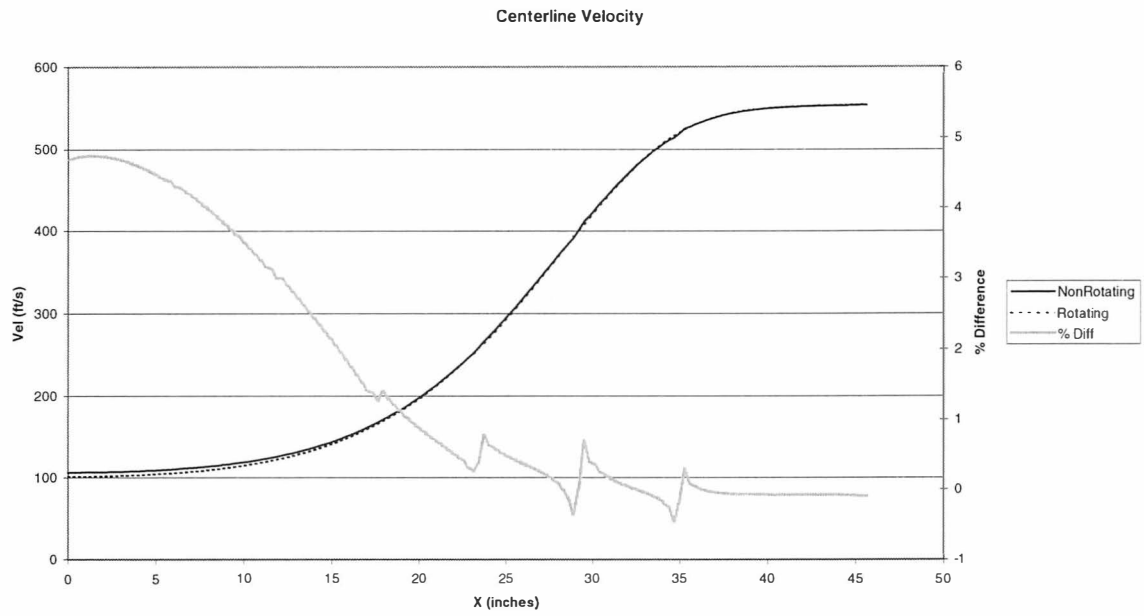


Figure 131. Comparison of Predicted Centerline Velocity for Corrected Rotating and Nonrotating Solution.

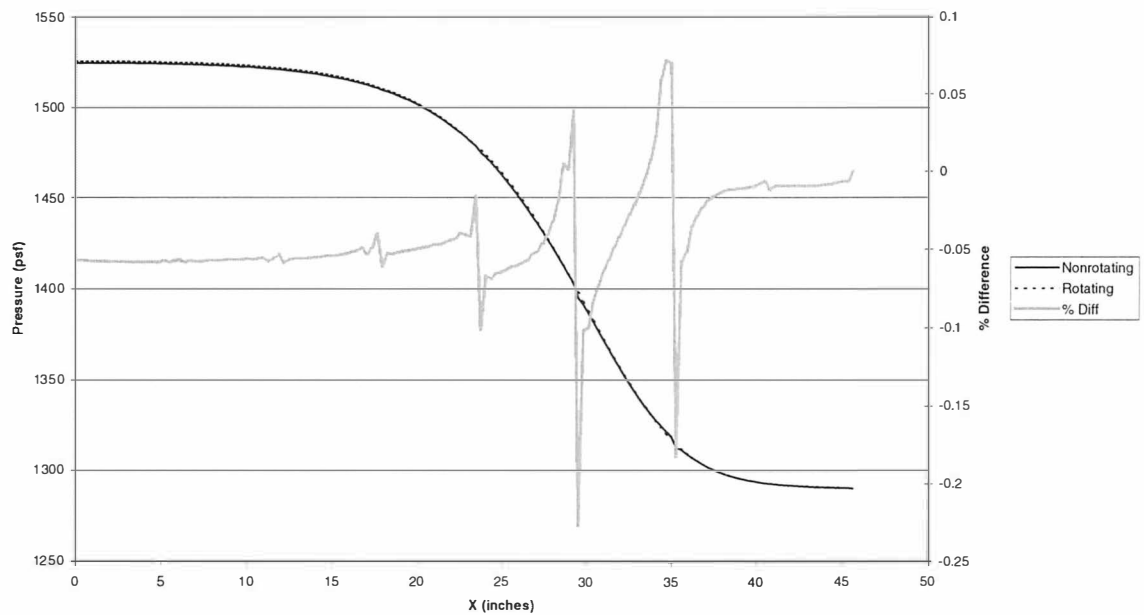


Figure 132. Comparison of Predicted Centerline Pressure for Corrected Rotating and Nonrotating Solution.

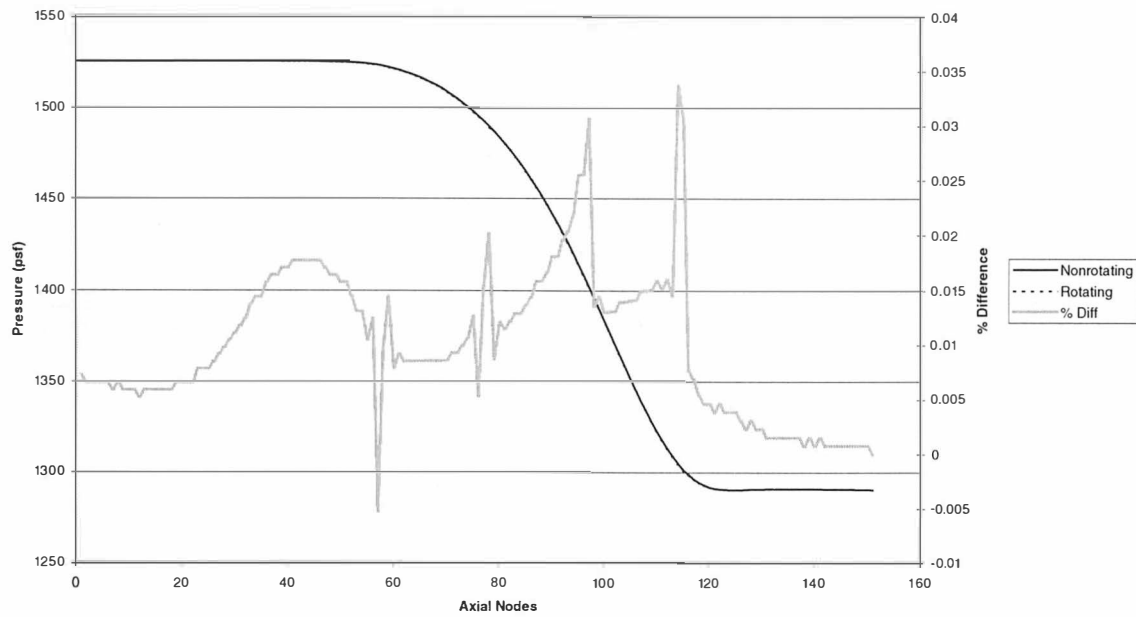


Figure 133. Comparison of Predicted Wall Pressure for Corrected Rotating and Nonrotating Solution.

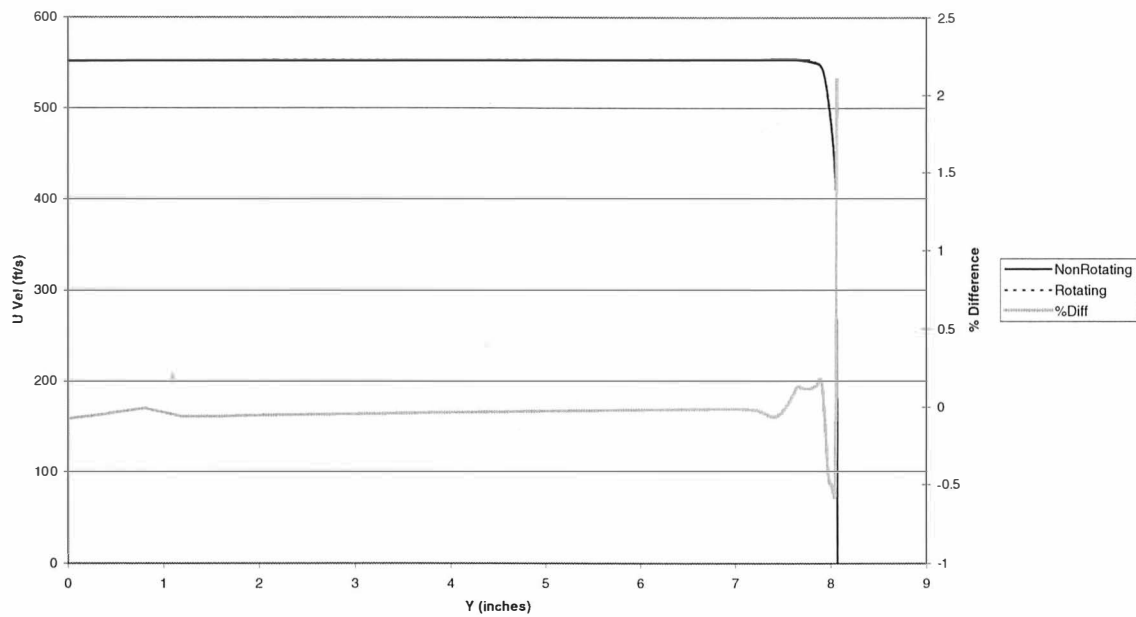


Figure 134. Comparison of Predicted Axial Component of Velocity at Station 8 for Corrected Rotating and Nonrotating Solution.

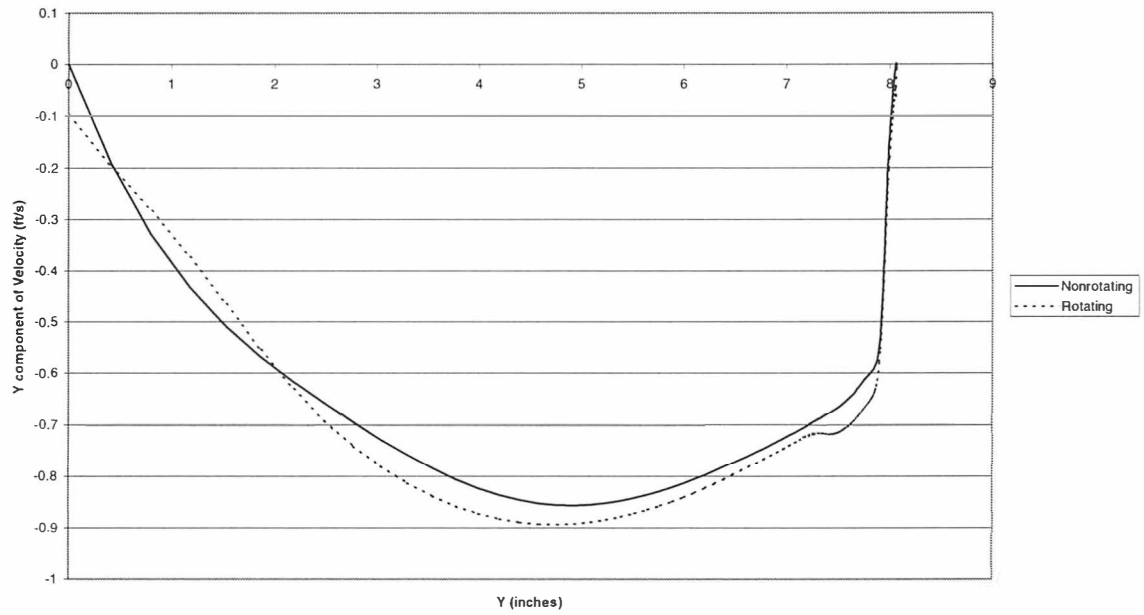


Figure 135. Comparison of Predicted Y Component of Velocity at Station 8 for Corrected Rotating and Nonrotating Solution.

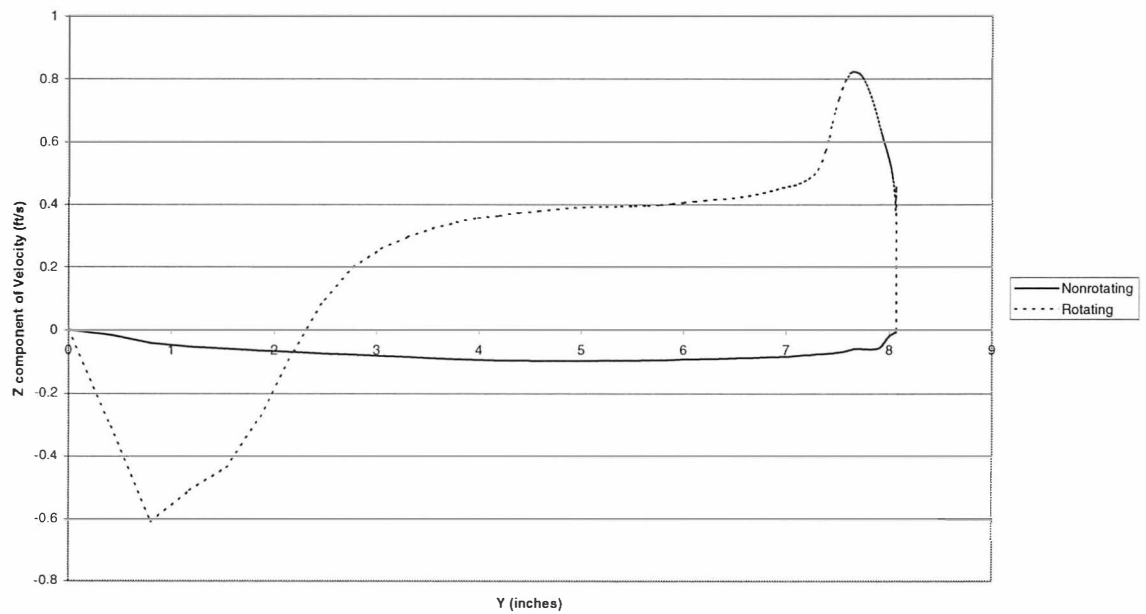


Figure 136. Comparison of Predicted Z Component of Velocity at Station 8 for Corrected Rotating and Nonrotating Solution.

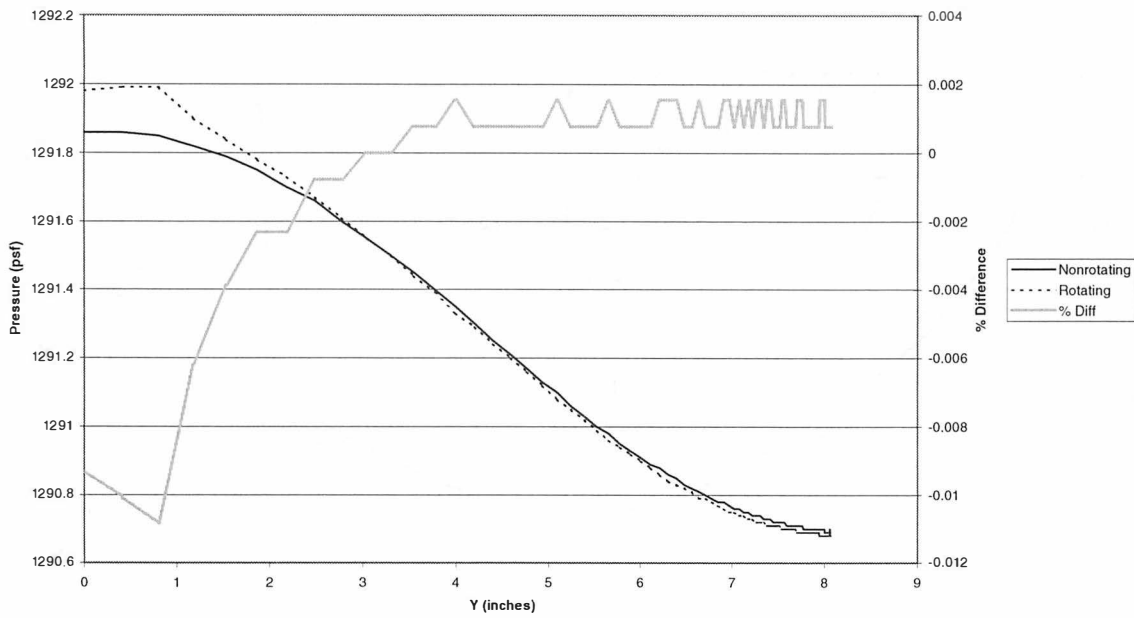


Figure 137. Comparison of Predicted Pressure at Station 8 for Corrected Rotating and Nonrotating Solution.

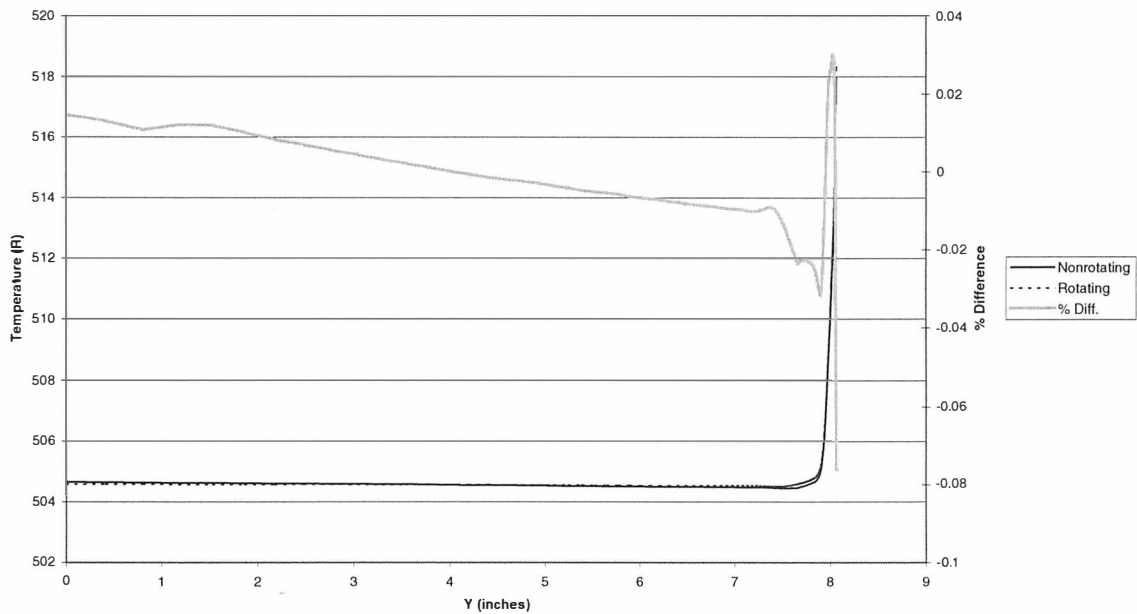


Figure 138. Comparison of Predicted Temperature at Station 8 for Corrected Rotating and Nonrotating Solution.

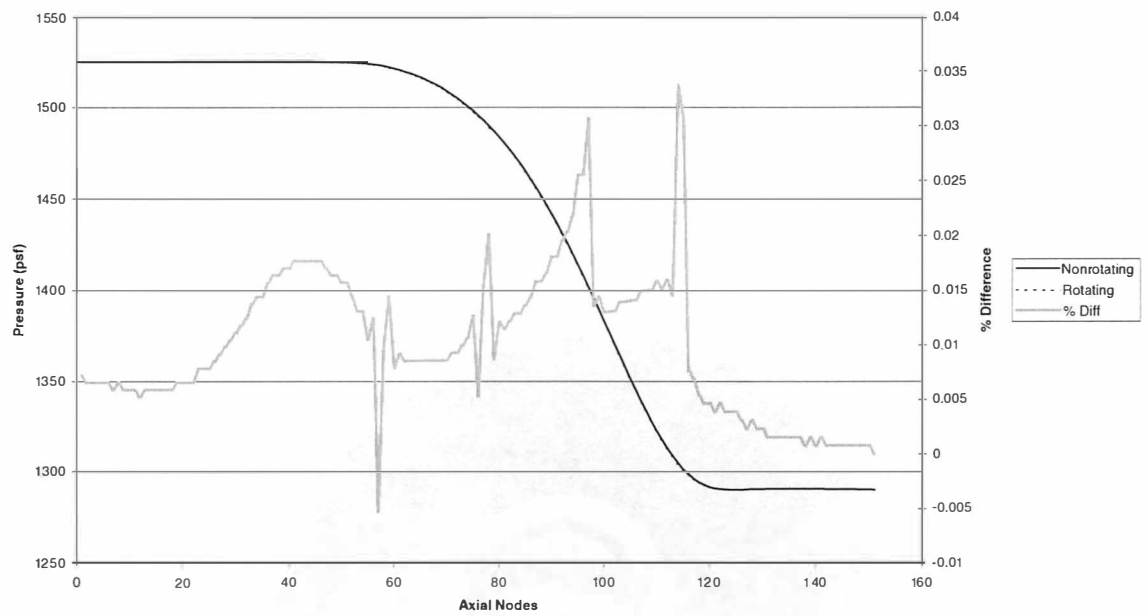


Figure 139. Predicted Pressure Along the $j=25$ Gridline for Corrected Rotating and Nonrotating Solution.

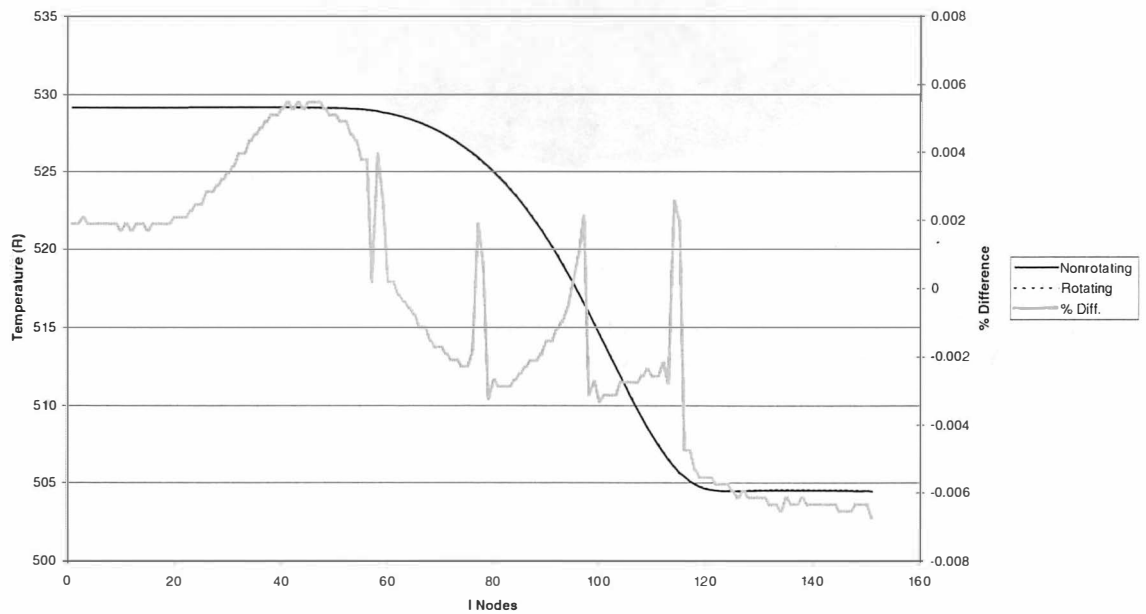
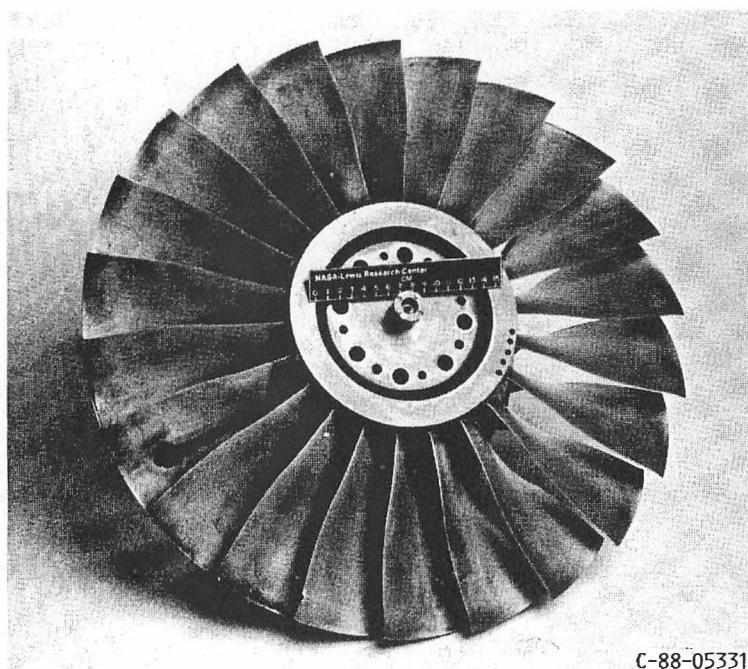


Figure 140. Predicted Temperature Along the $j=25$ Gridline for Corrected Rotating and Nonrotating Solution.



C-88-05331

Figure 141. Rotor 67.

Source: Strazisar, A. J., Wood, J.R., Hathaway, M.D., and Suder, K.L., "Laser Anemometer Measurements in a Transonic Axial-Flow Ran Rotor," NASA Technical Paper 2879, November 1989.

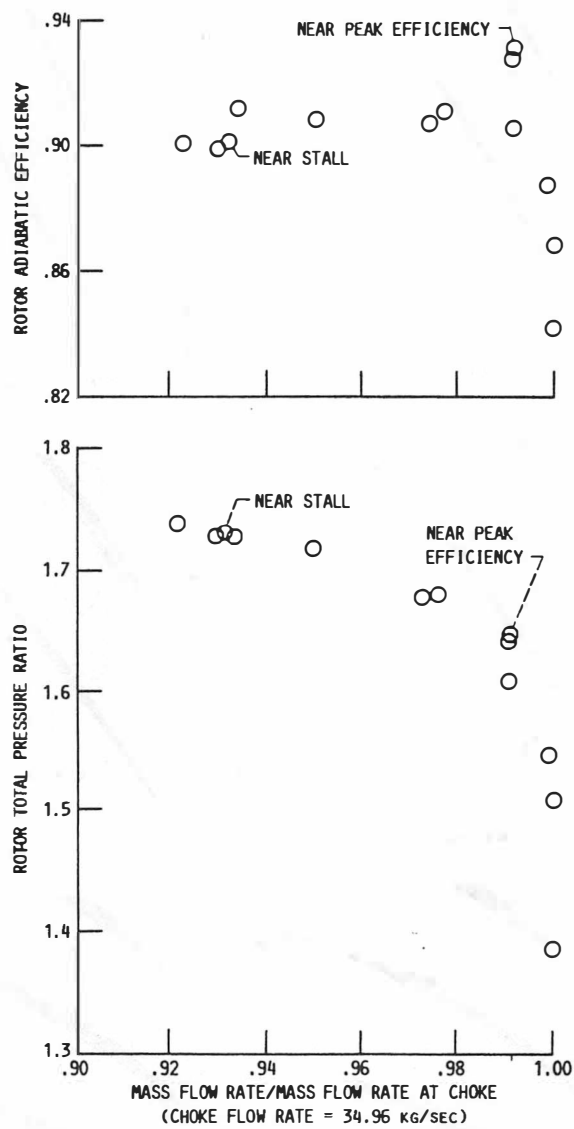


Figure 142. Rotor 67 Pressure Ratio Performance.

Source: Strazisar, A. J., Wood, J.R., Hathaway, M.D., and Suder, K.L., "Laser Anemometer Measurements in a Transonic Axial-Flow Ran Rotor," NASA Technical Paper 2879, November 1989.

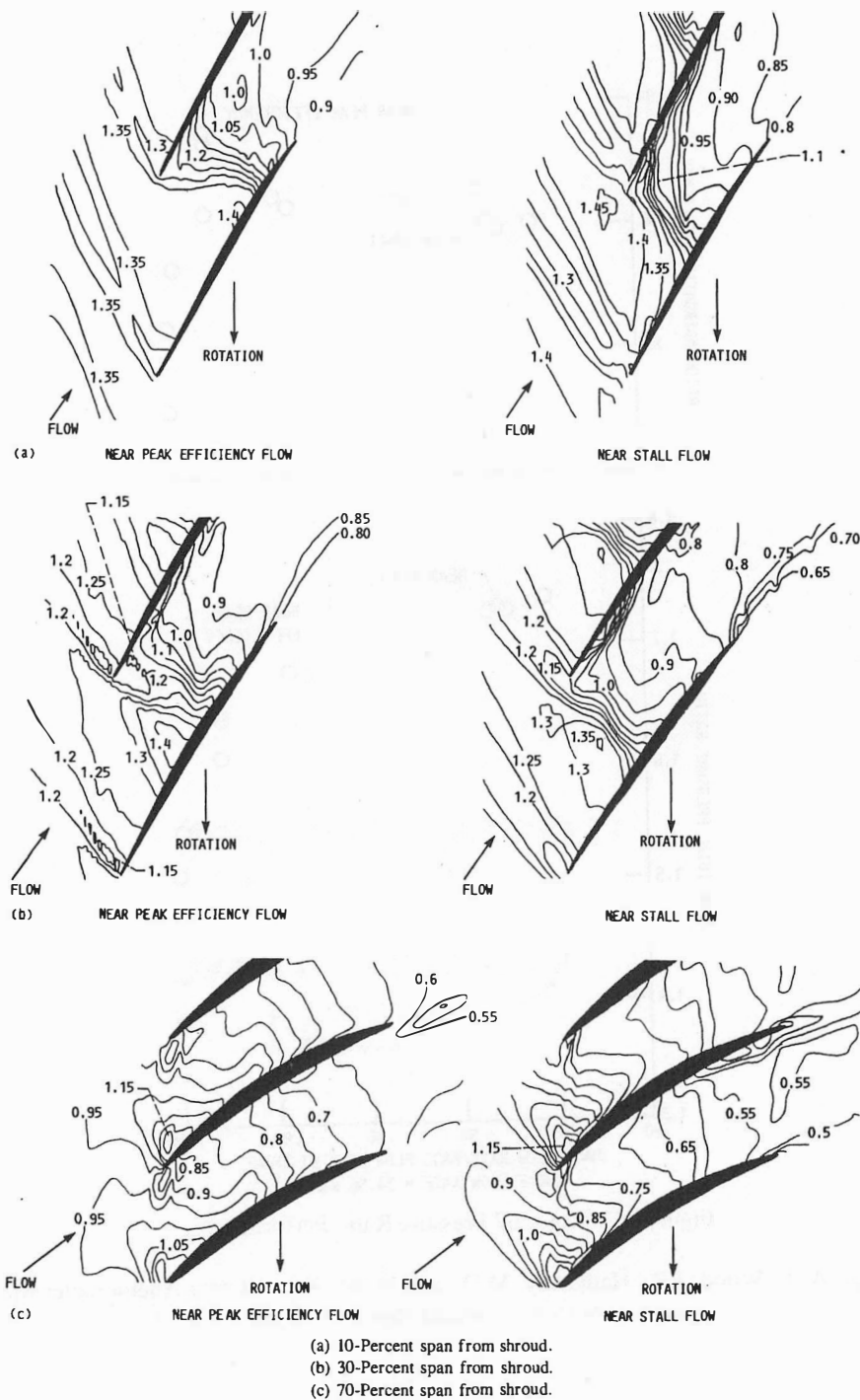


Figure 143. Contour Plots of Rotor 67 Relative Mach Number.

Source: Strazisar, A. J., Wood, J.R., Hathaway, M.D., and Suder, K.L., "Laser Anemometer Measurements in a Transonic Axial-Flow Ran Rotor," NASA Technical Paper 2879, November 1989.

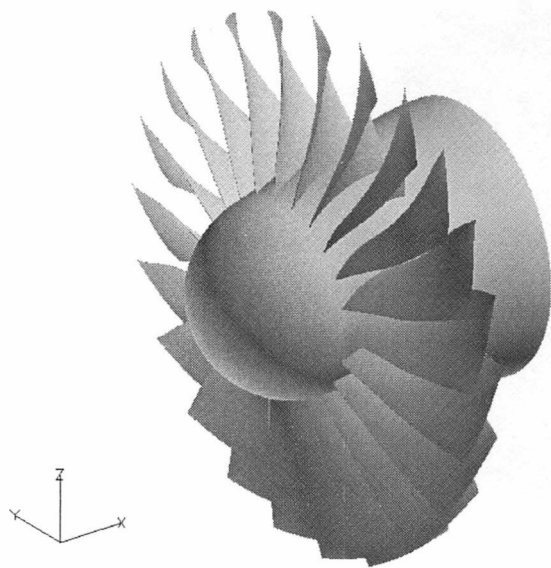


Figure 144. Model of Rotor 67 with Spherical Nose.

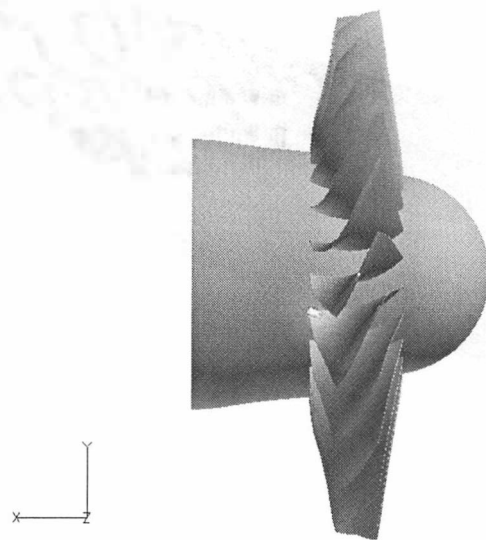


Figure 145. Side View of Rotor 67 Model.

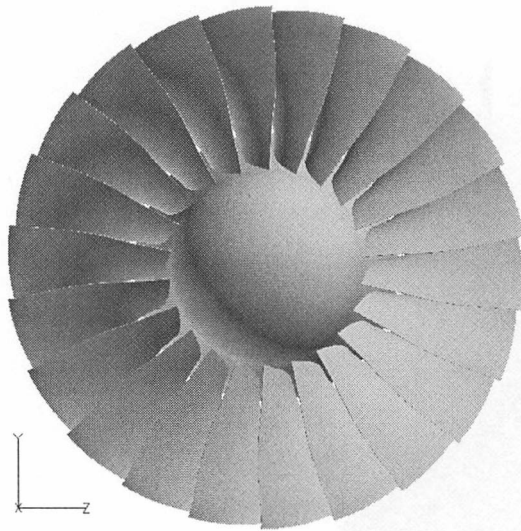


Figure 146. Front View of Rotor 67 Model.

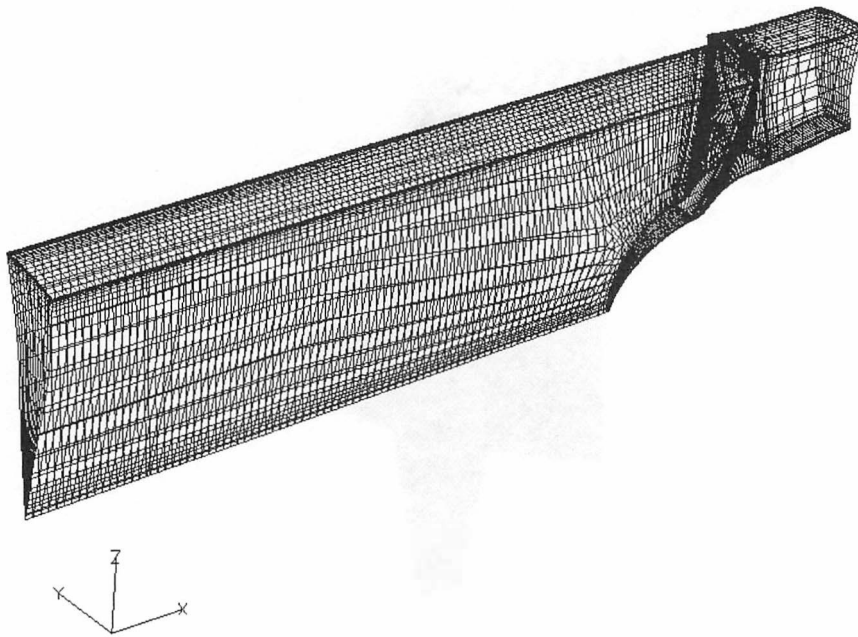


Figure 147. Grid of Fan with Straight Inlet Duct.

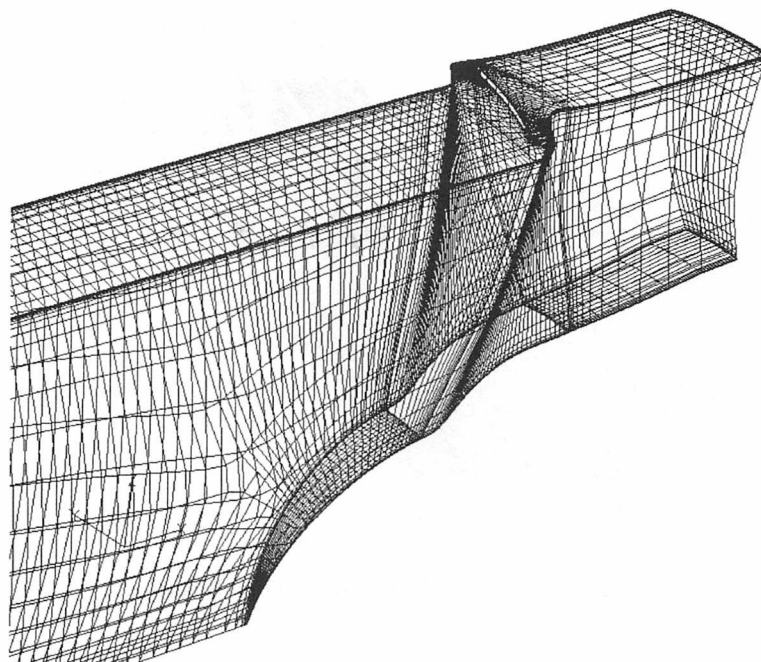


Figure 148. Close Up of Grid in the Region of the Fan.

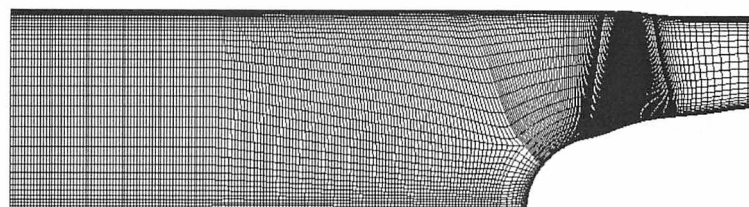


Figure 149. Side View of Grid: Fan with Straight Duct.

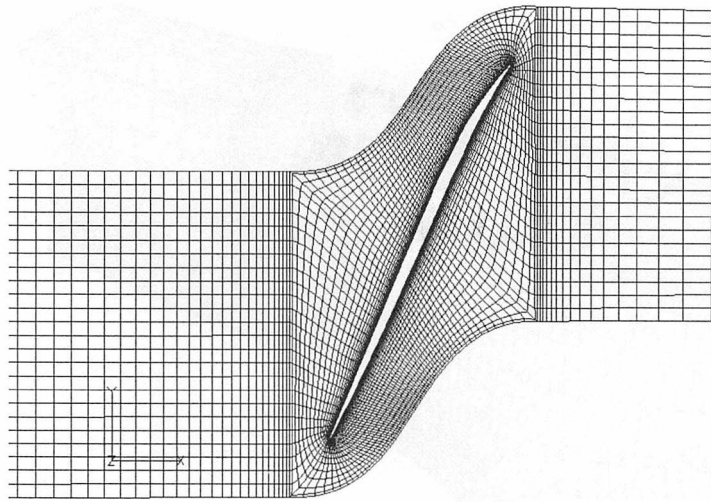


Figure 150. Top View of Grid Around Fan Blade.

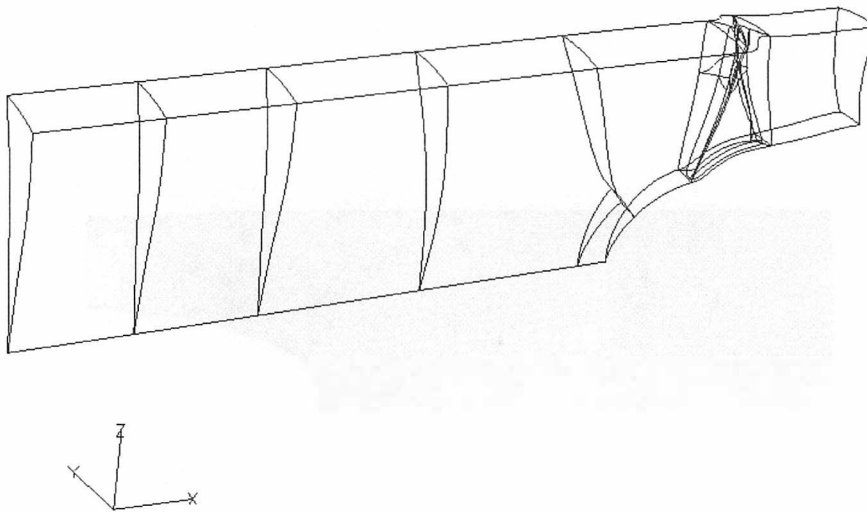


Figure 151. Grid Computational Zones for Fan in a Straight Duct

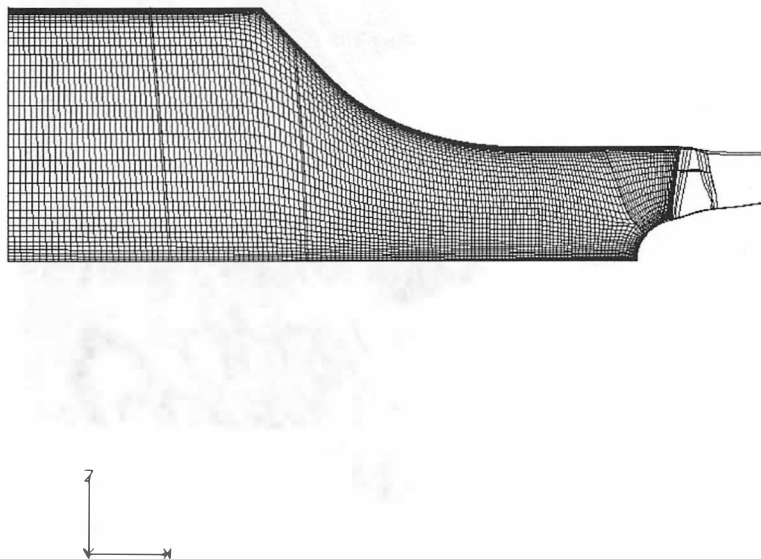


Figure 152. Side View of Grid: Fan with Bellmouth Inlet.

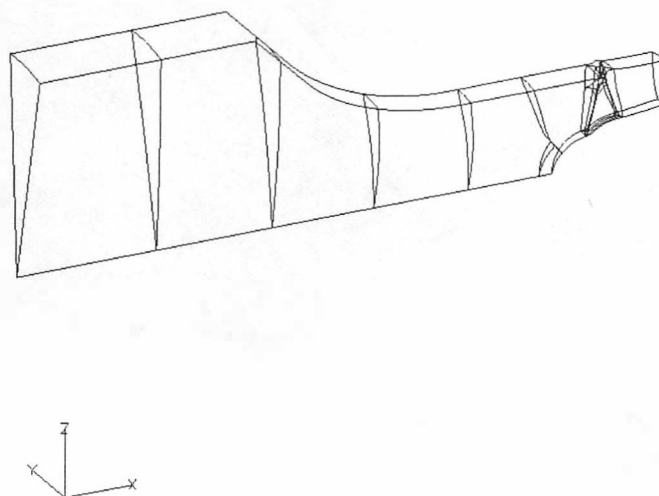


Figure 153. Computational Zones for Fan with Bellmouth Inlet.

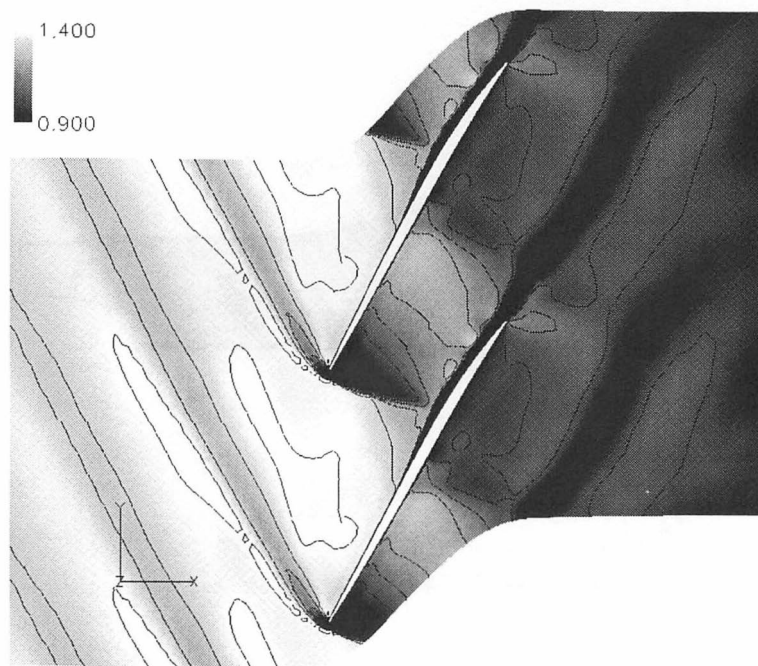


Figure 154. Computed Mach Contours At 10% of Span From Tip.

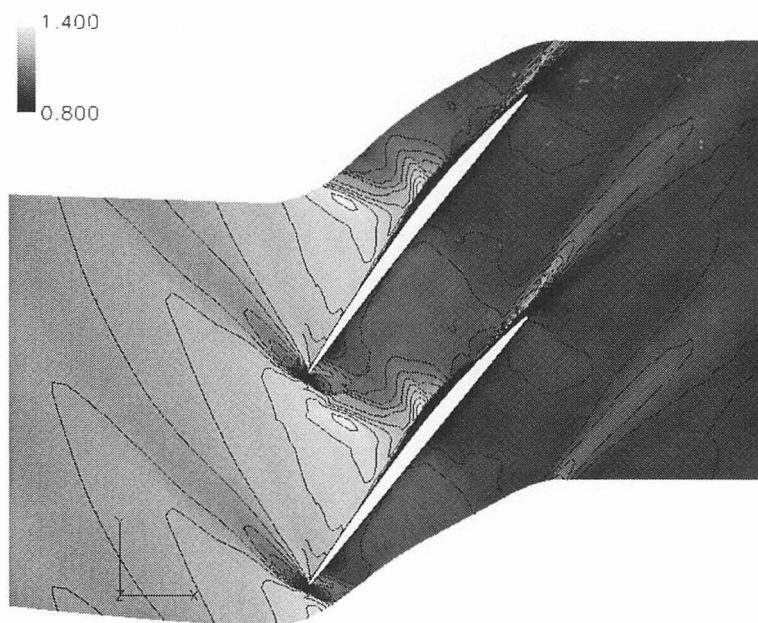


Figure 155. Computed Mach Contours At 30% of Span From Tip.

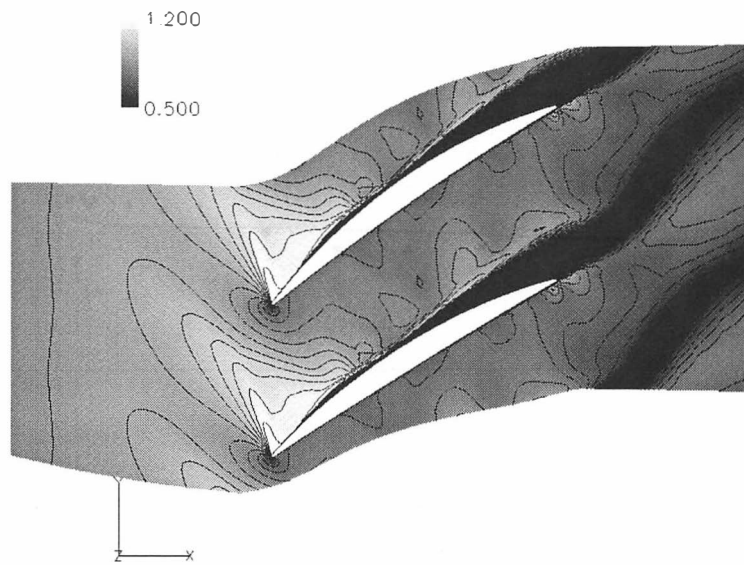


Figure 156. Computed Mach Contours At 70% of Span From Tip.

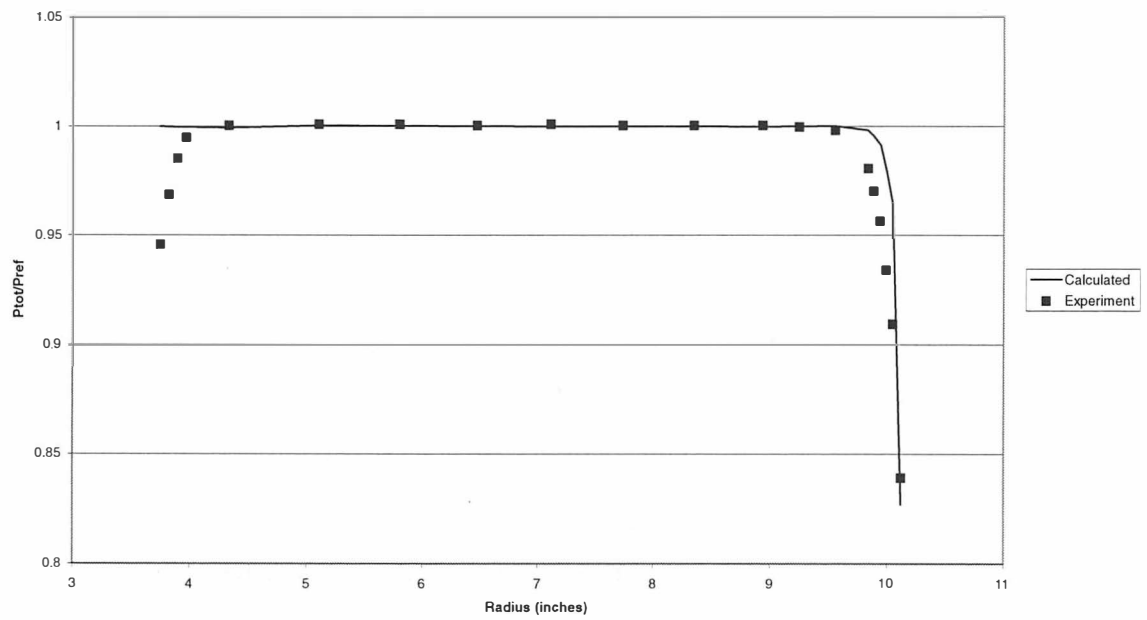


Figure 157. Comparison of Experimental and Computed Total Pressure At Aerodynamic Station 1.

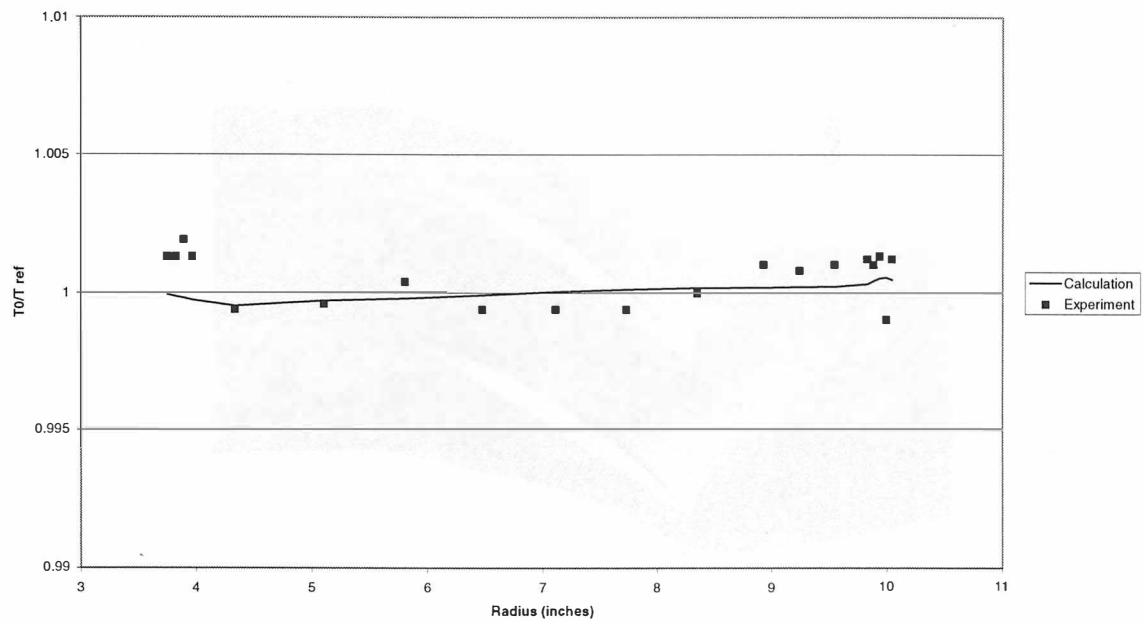


Figure 158. Comparison of Experimental and Computed Total Temperature at Aerodynamic Station 1.

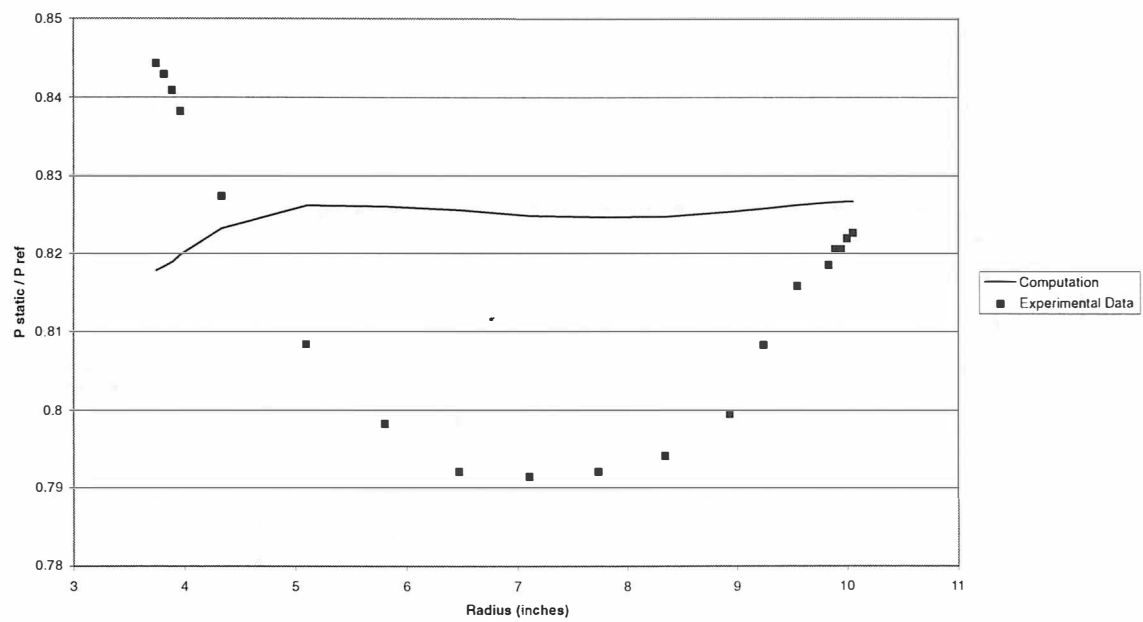


Figure 159. Comparison of Experimental and Computed Static Pressure at Aerodynamic Station 1.

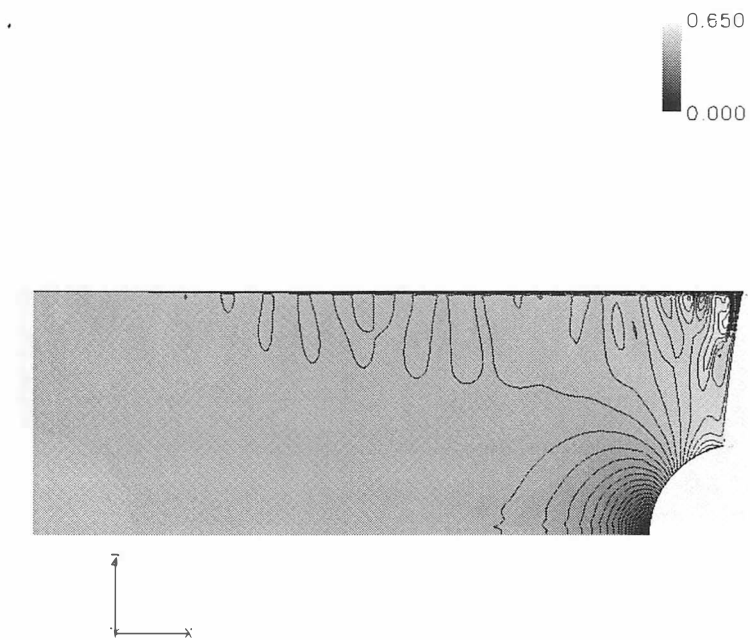


Figure 160. Predicted Contours of Mach Number in a Constant Area Inlet Duct.

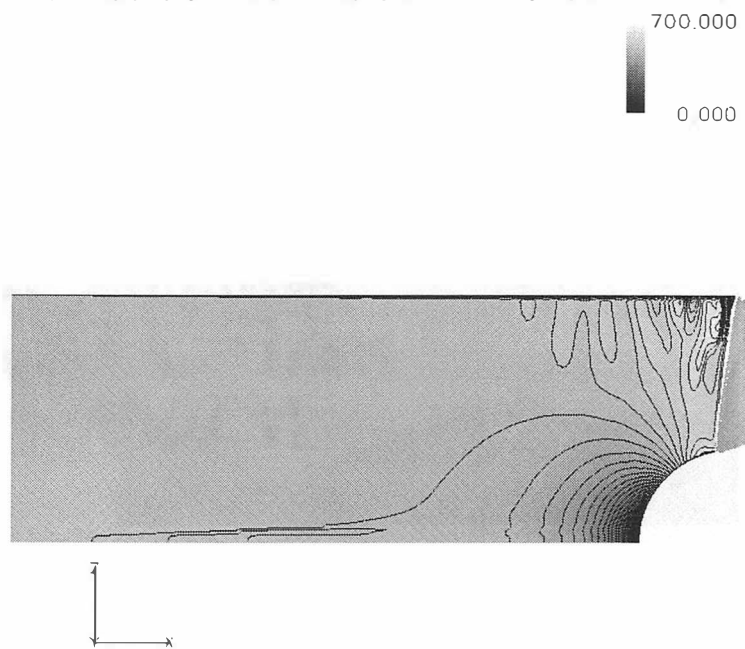


Figure 161. Predicted Contours of X-Component of Velocity in a Constant Area Inlet Duct.

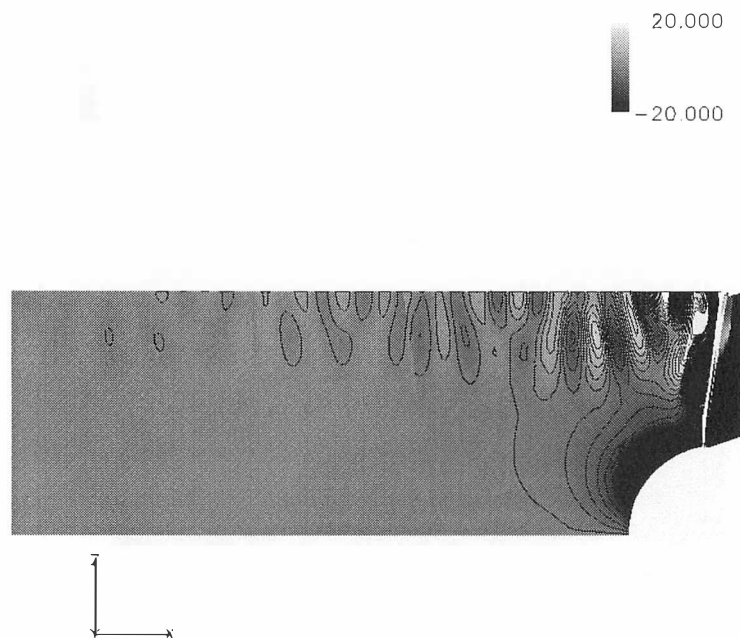


Figure 162. Predicted Contours of Y-Component of Velocity in a Constant Area Inlet Duct.

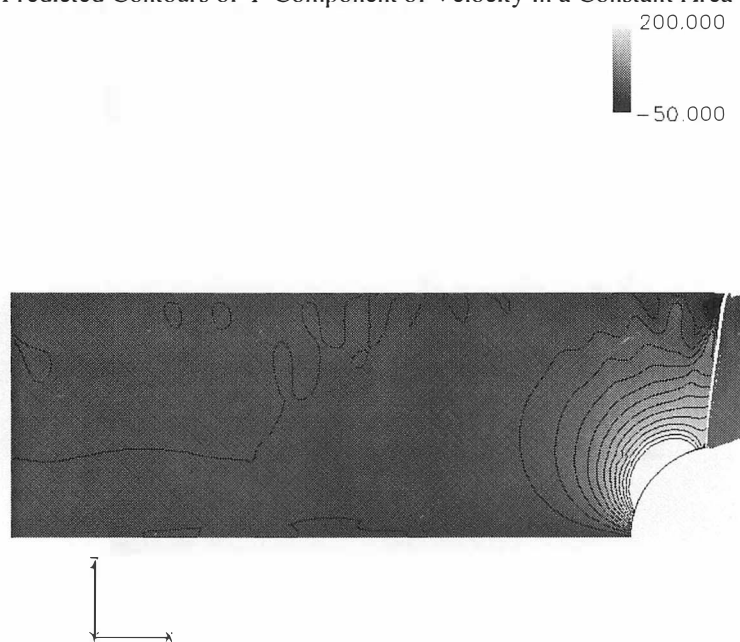


Figure 163. Predicted Contours of Z-Component of Velocity in a Constant Area Inlet Duct.

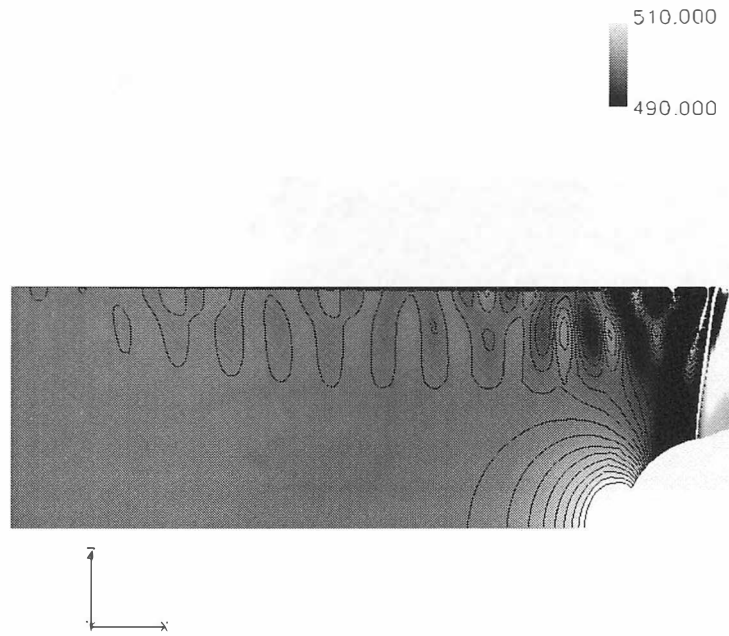


Figure 164. Predicted Contours of Static Temperature in a Constant Area Inlet Duct.

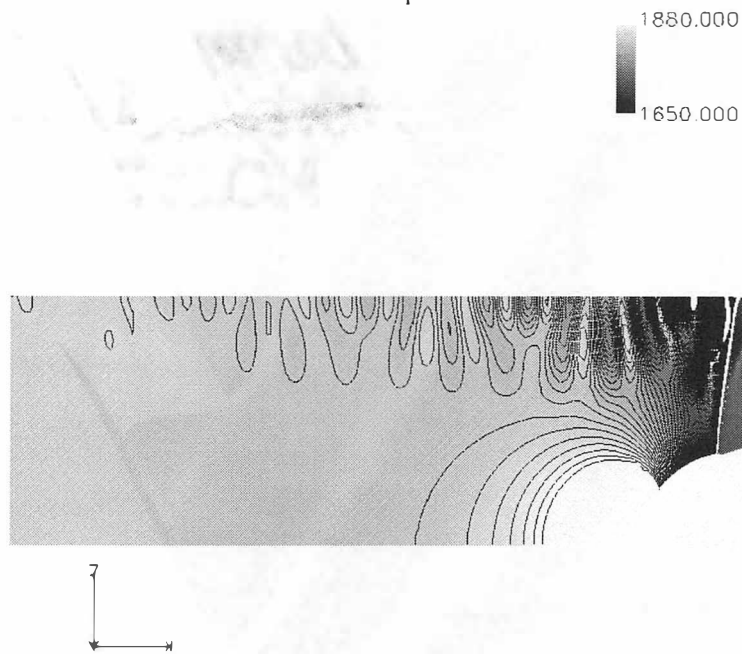


Figure 165. Predicted Contours of Static Pressure in a Constant Area Inlet Duct.

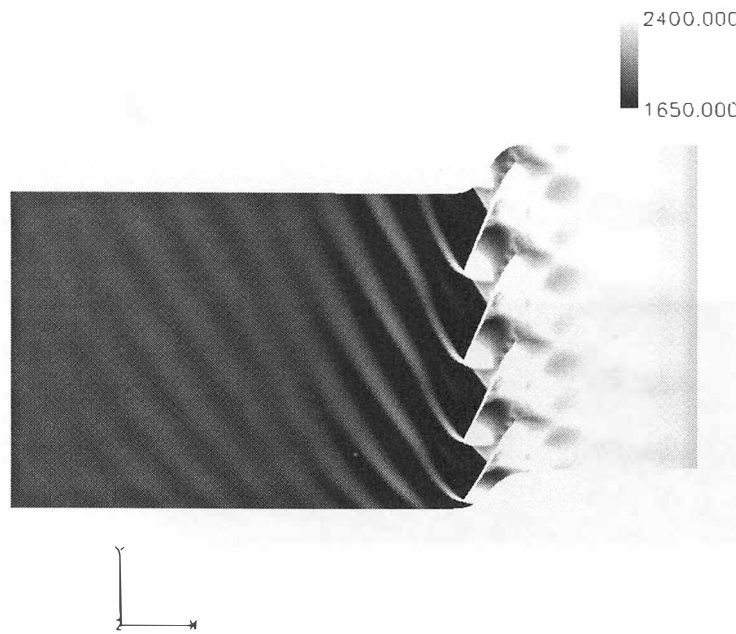


Figure 166. Predicted Static Pressure Field at Wall in the Region of the Fan.



Figure 167. Details of Shock at Fan Blade Tips.

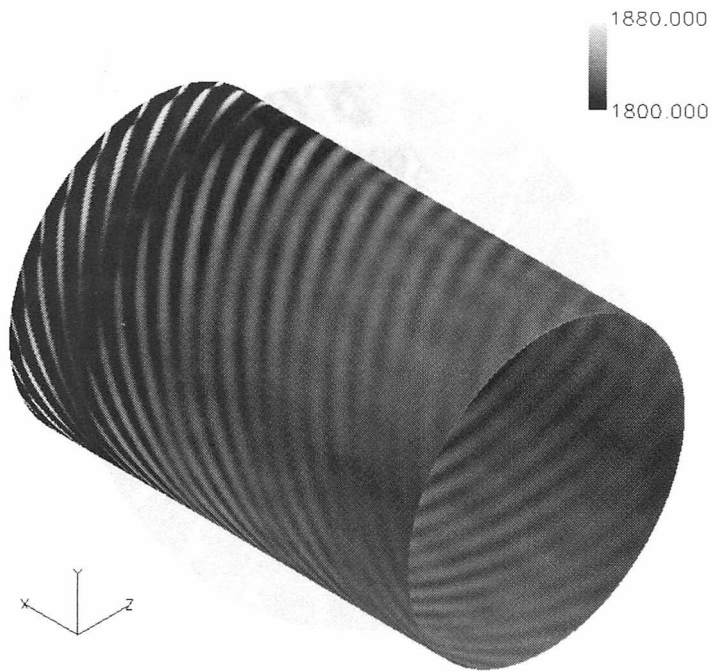


Figure 168. Three-Dimensional View of Static Pressure on Duct Wall.

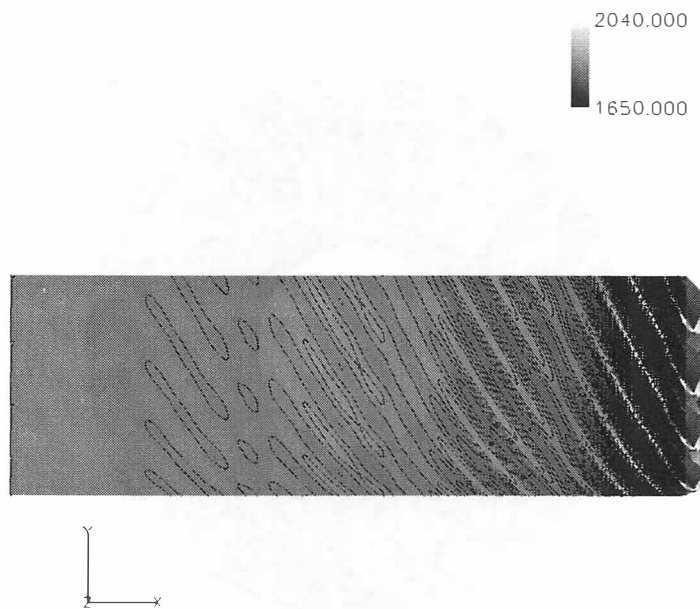


Figure 169. Detail of Static Pressure on Duct Wall.

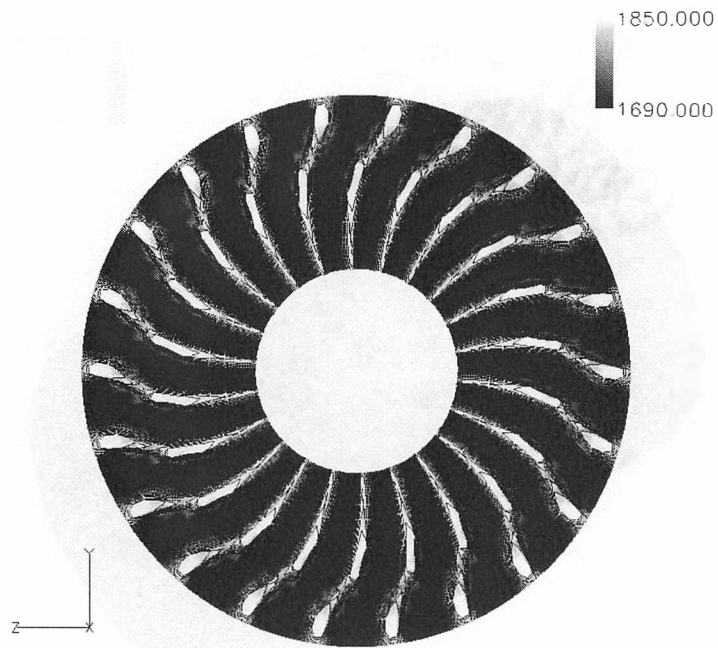


Figure 170. Predicted Pressure Contours at $x = -0.2$.

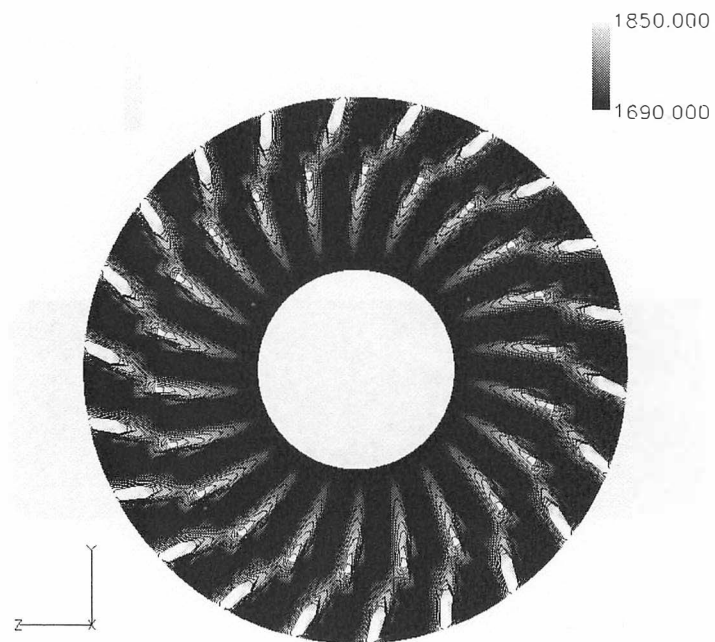


Figure 171. Predicted Pressure Contours at $x = -0.5$.

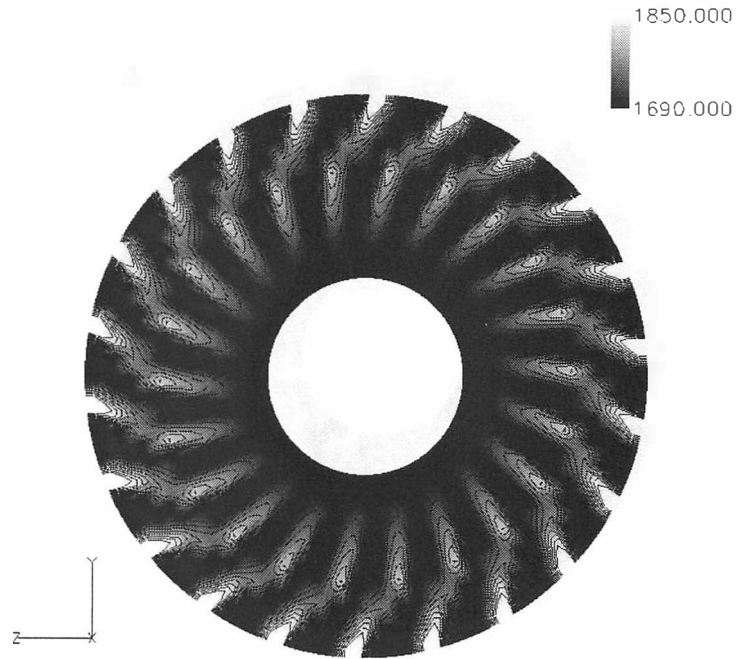


Figure 172. Predicted Pressure Contours at $x = -1$.

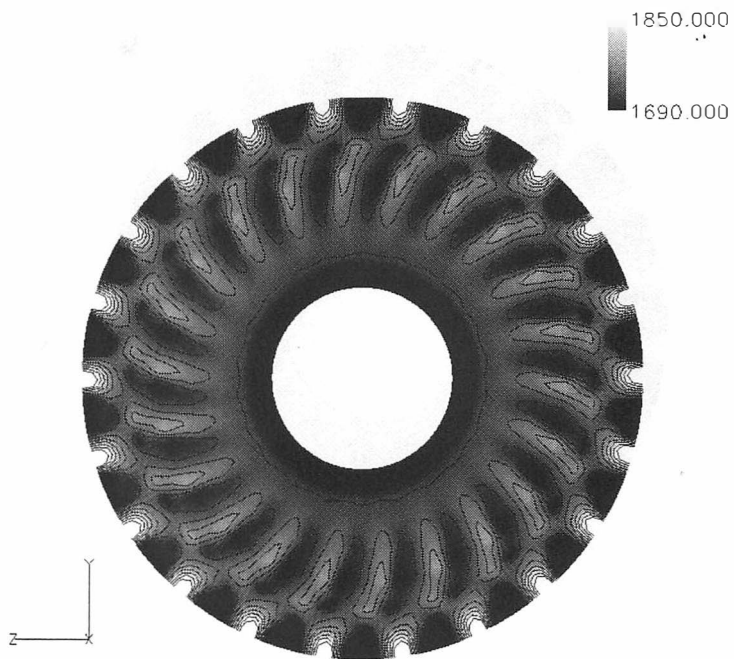


Figure 173. Predicted Pressure Contours at $x = -1.5$.

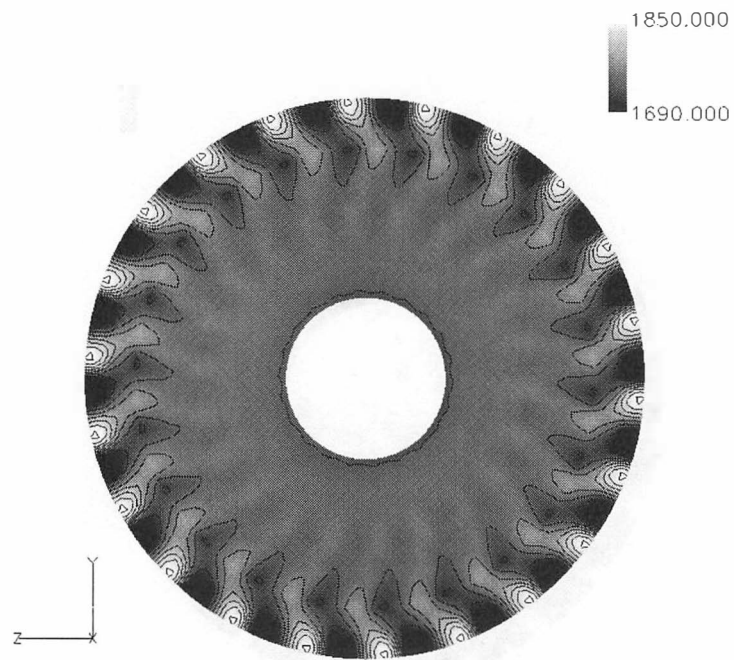


Figure 174. Predicted Pressure Contours at $x = -2$.

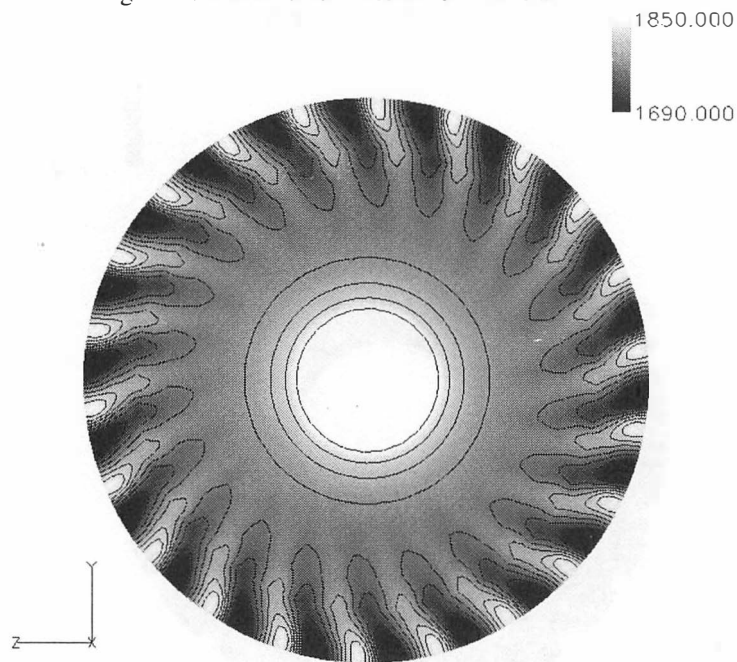


Figure 175. Predicted Pressure Contours at $x = -2.5$.

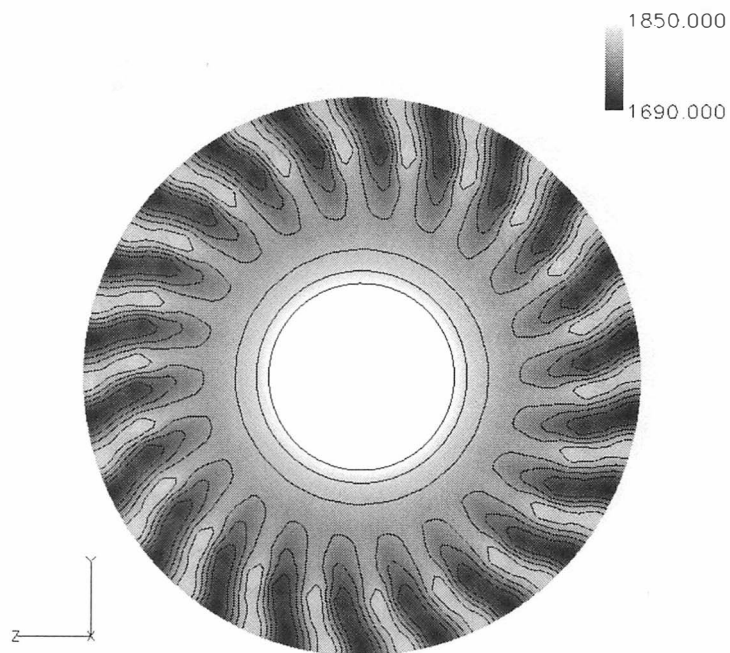


Figure 176. Predicted Pressure Contours at $x = -3$.

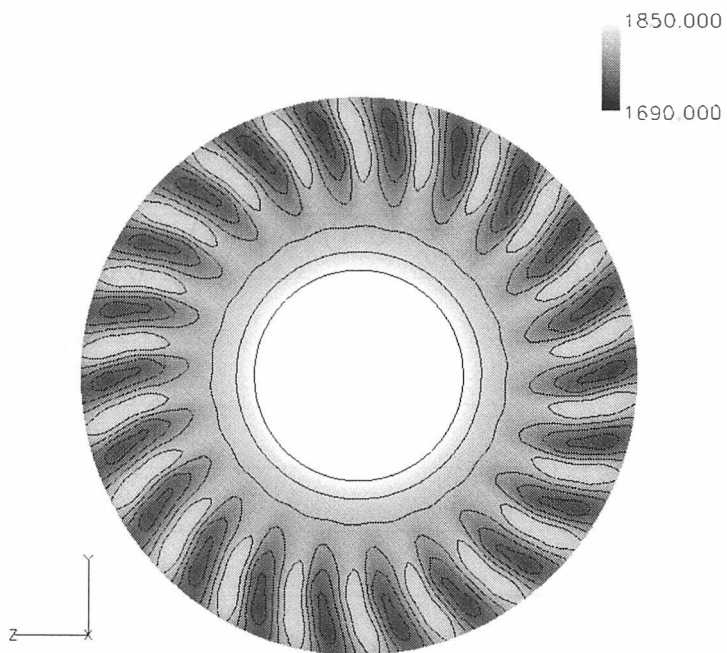


Figure 177. Predicted Pressure Contours at $x = -3.5$.

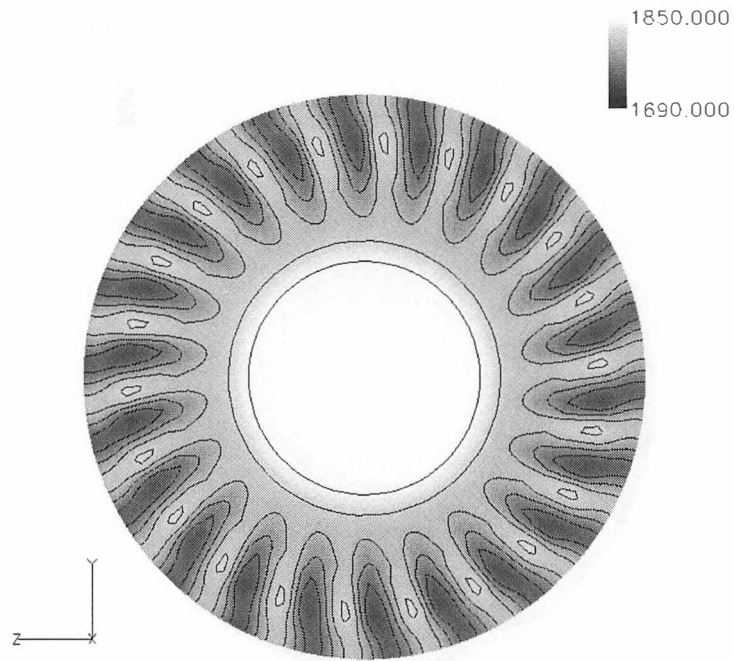


Figure 178. Predicted Pressure Contours at $x = -4$.

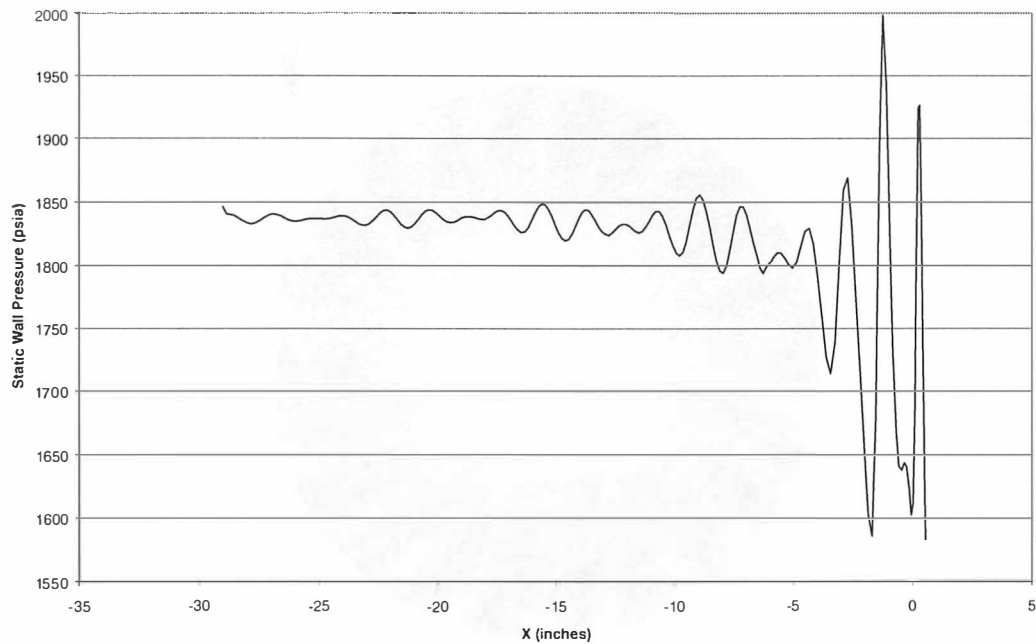


Figure 179. Instantaneous Static Pressure On Wall vs. X-Coordinate.

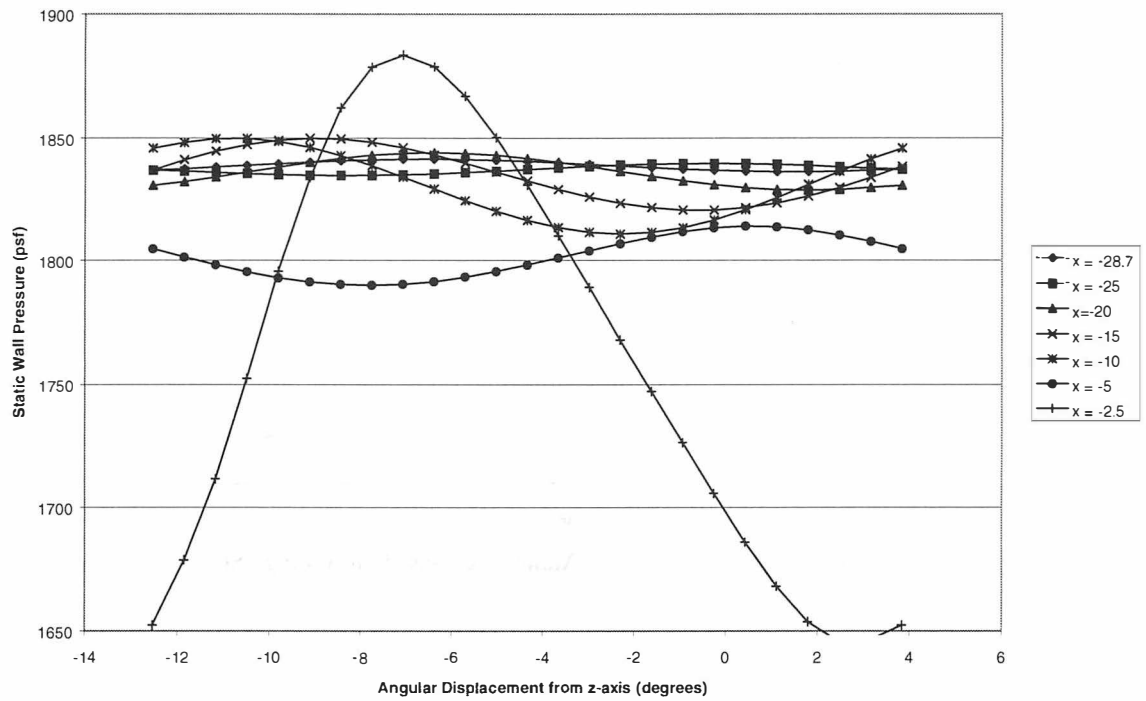


Figure 180. Predicted Distributions of Instantaneous Static Pressure On Wall At Constant X Plane.

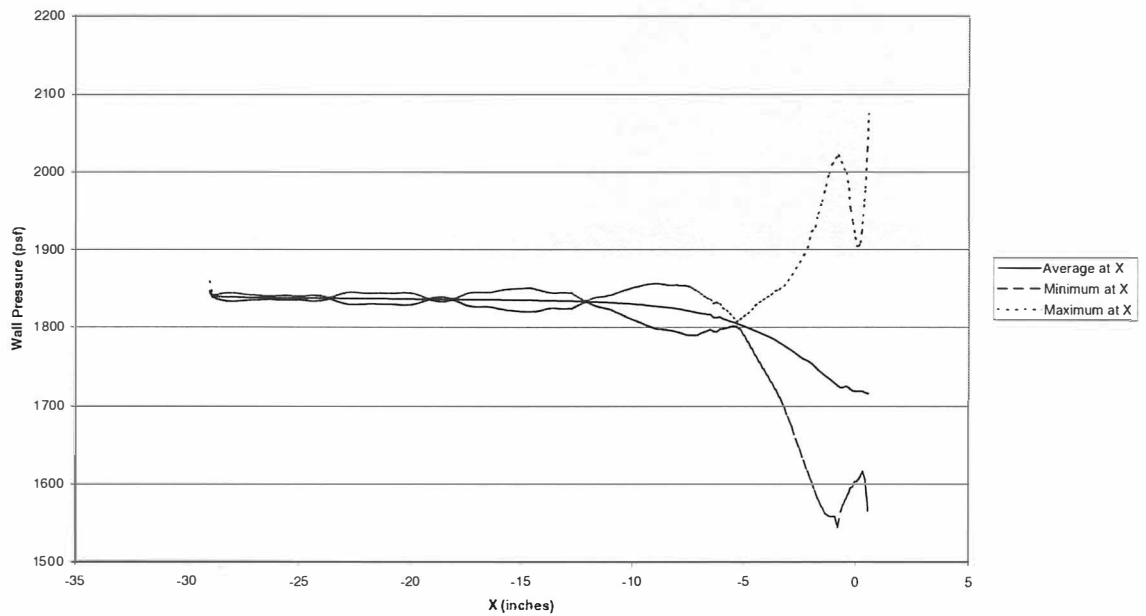


Figure 181. Max, Min, and Average Static Pressure on Wall vs. X-Coordinate.

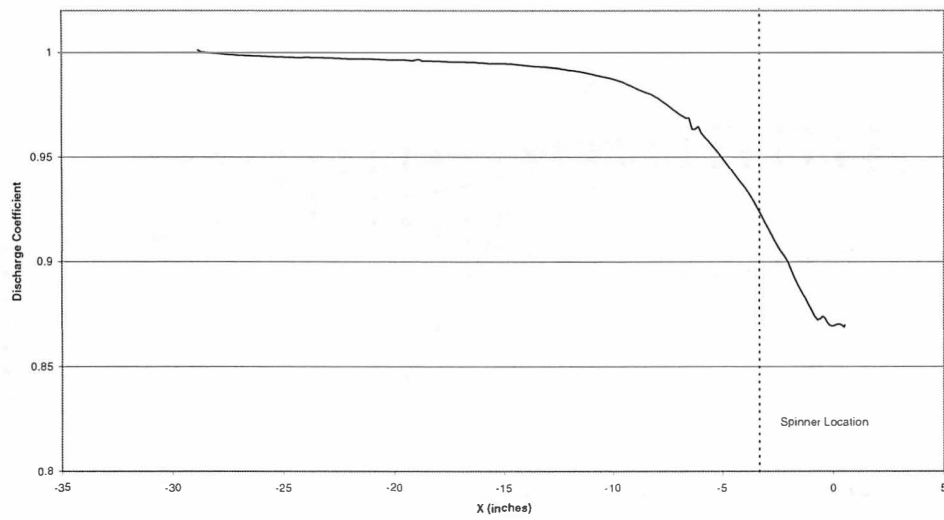


Figure 182. Computed Discharge Coefficient Along Duct Based on Average Static Pressure.

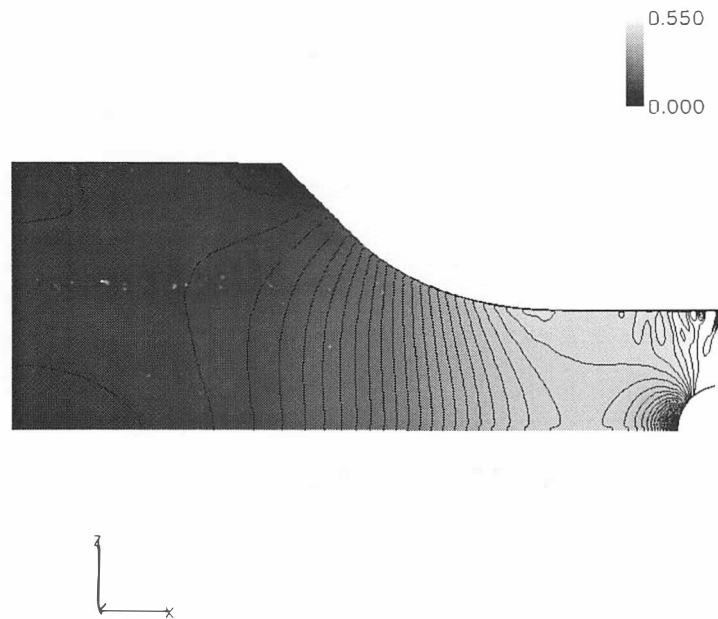


Figure. 183. Predicted Mach Number Contours in Bellmouth Upstream of Fan.

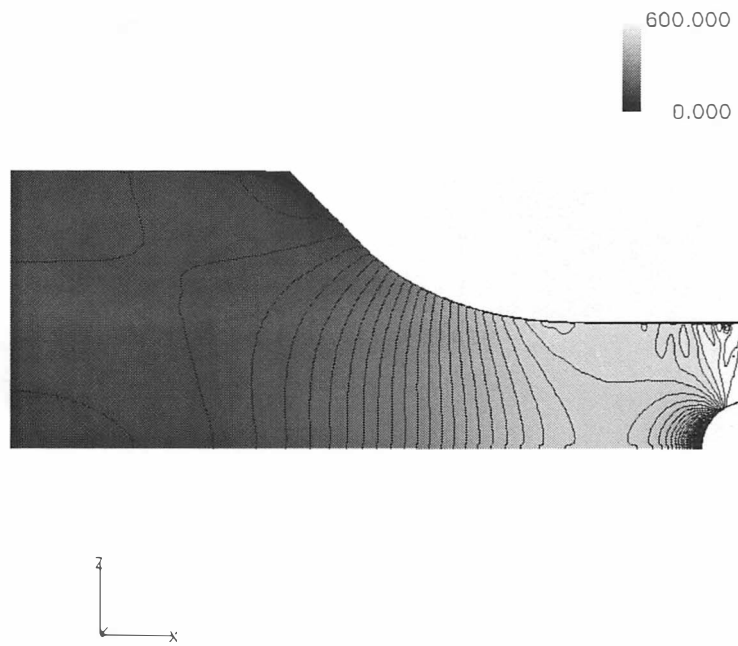


Figure 184. Predicted X-Component of Velocity Contours in Bellmouth Upstream of Fan.

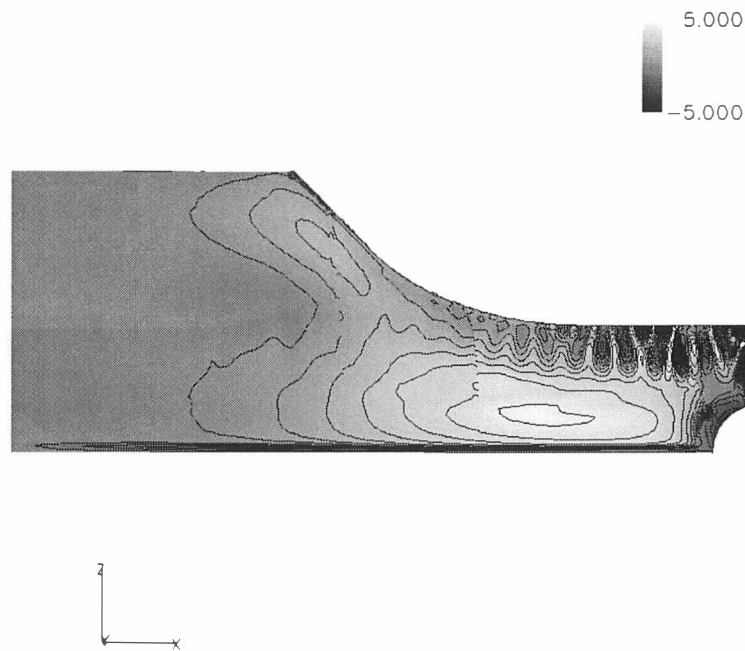


Figure 185. Predicted Y-Component of Velocity Contours in Bellmouth Upstream of Fan.

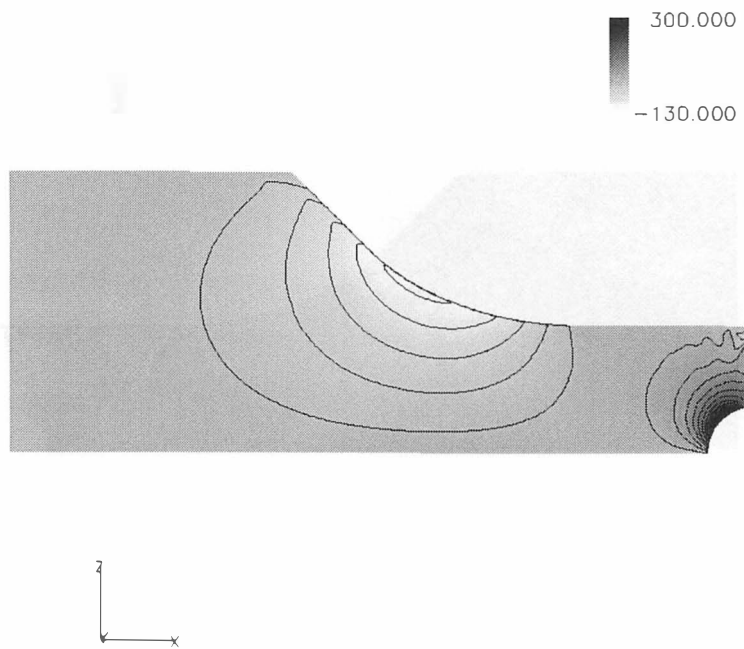


Figure 186. Predicted Z-Component of Velocity Contours in Bellmouth Upstream of Fan.

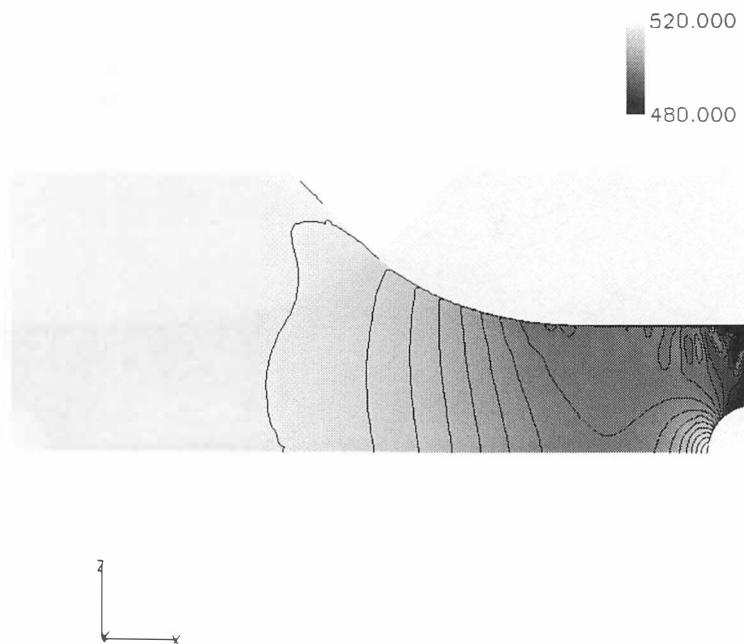


Figure 187. Predicted Temperature Contours in Bellmouth Upstream of Fan.

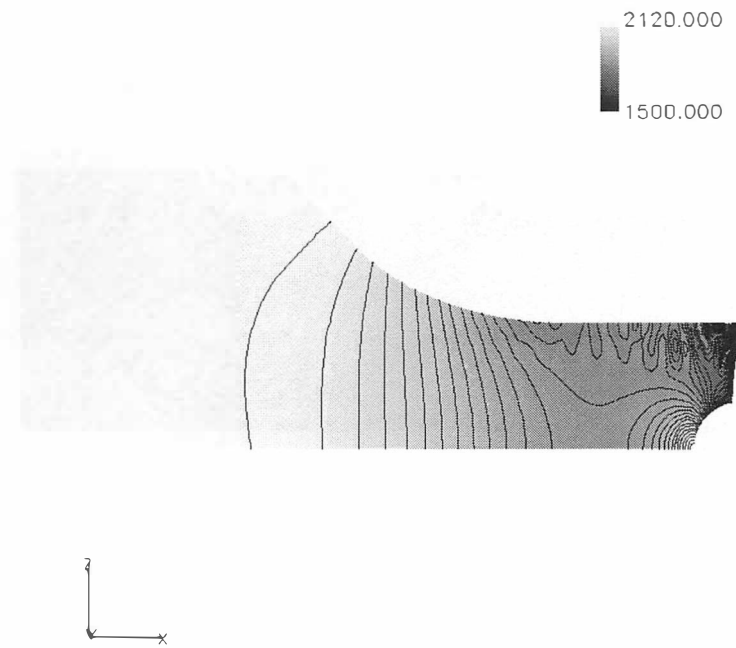


Figure 188. Predicted Static Pressure Contours in Bellmouth Upstream of Fan.

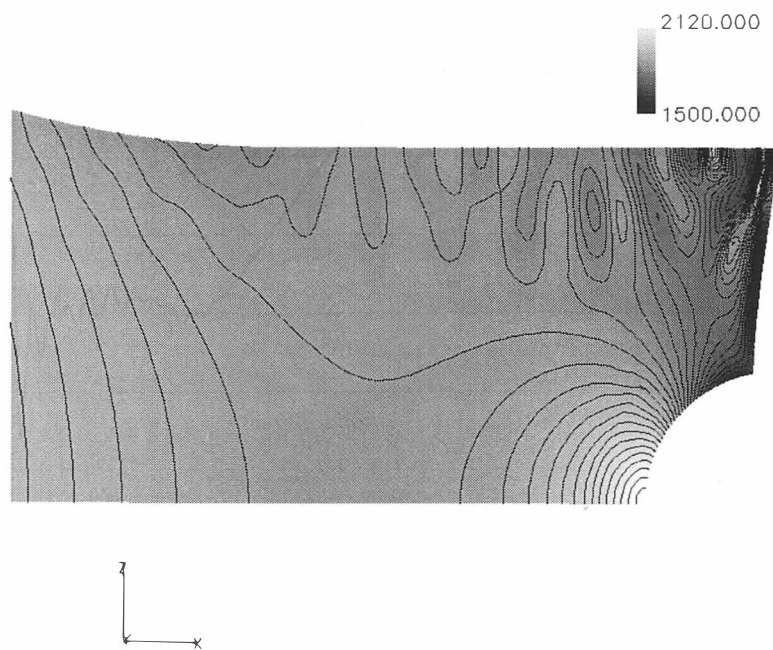


Figure 189. Predicted Static Pressure Contours in Bellmouth Throat Region.

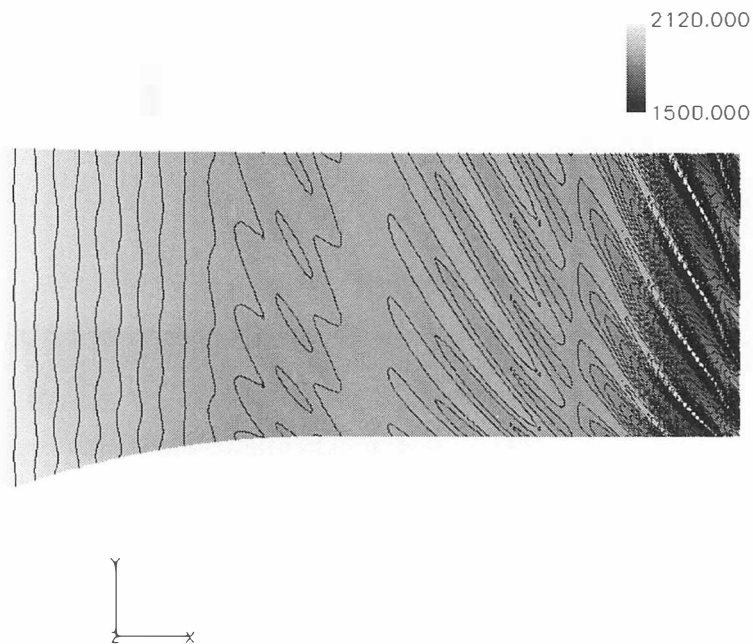


Figure 190. Predicted Static Pressure Contours at the Wall in Throat Region of Bellmouth.

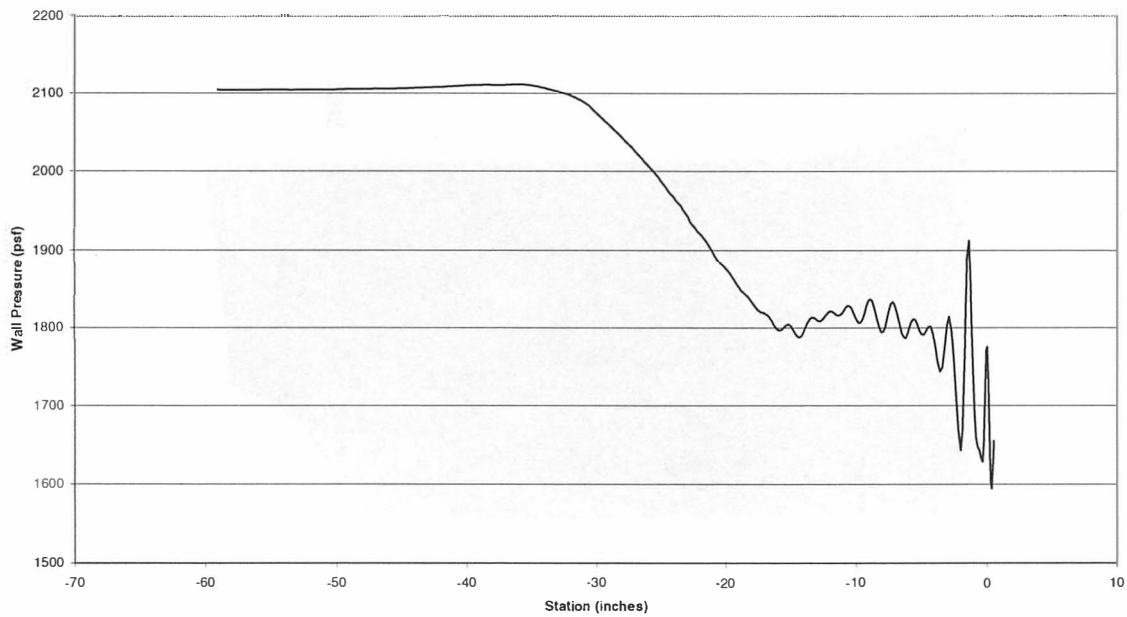


Figure 191. Instantaneous Wall Pressure Along Axis of Bellmouth.

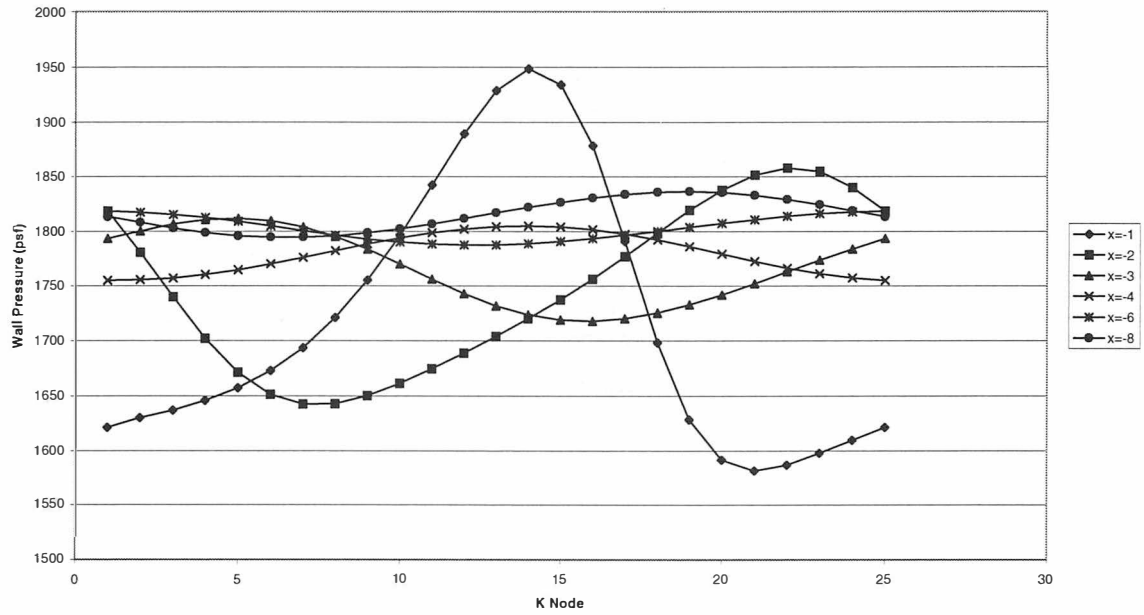


Figure 192. Wall Static Pressure at Constant Axial Planes in Throat Region.

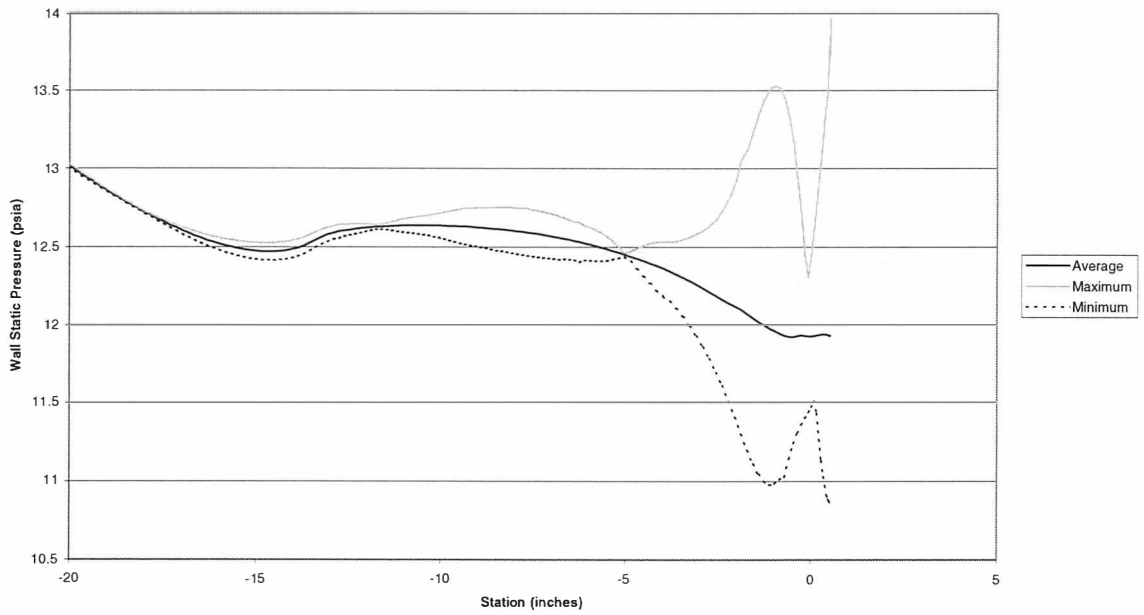


Figure 193. Maximum, Minimum, and Average Wall Pressure Along Axis of Bellmouth.

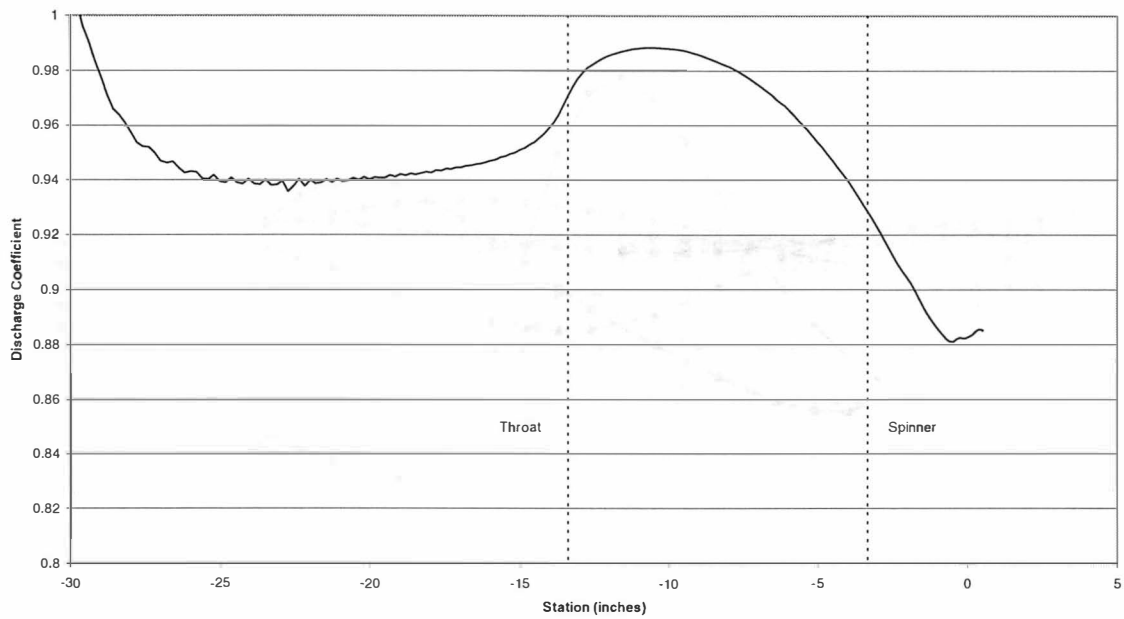


Figure 194. Computed Discharge Coefficient Along Axis of Bellmouth with Fan.

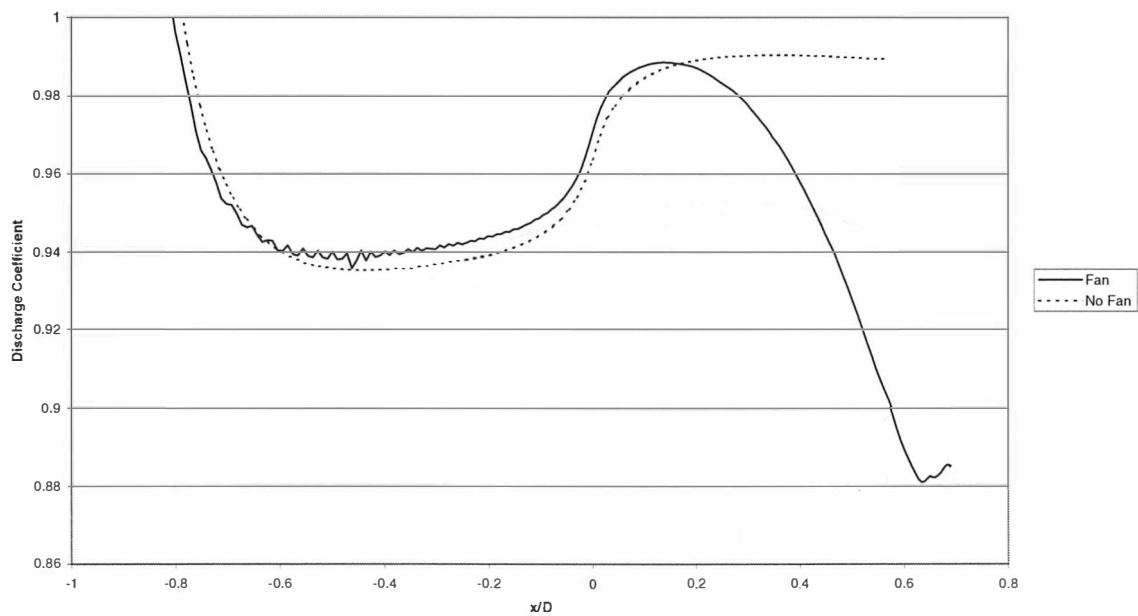


Figure 195. Comparison of Computed Discharge Coefficient in Bellmouth With Fan and Without Fan.

Vita

Charles Lynn Sebourn was born in Blytheville, Arkansas, on July 27th, 1966. He attended elementary school in Blytheville, Arkansas, and North Pemiscot County Schools in Pemiscot County, Missouri. He also attended Junior High Schools at North Pemiscot Junior High School. He graduated from Doniphan High School, Doniphan, Missouri, in 1984. In the fall of 1984 he entered the University of Missouri-Rolla and in May of 1989 he graduated with a Bachelor of Science degree in Mechanical Engineering. In the fall of 1990 he enrolled at the University of Tennessee Space Institute. He graduated with a Master of Science degree in Aerospace Engineering in July 1992. In August of 1992 he accepted employment with Sverdrup Technology, Inc. at the Arnold Engineering Development Center in facility analysis. In August of 1997, he was the recipient of the Resident Graduate Study Program Scholarship which permitted him to attend school full time for one year while still receiving salary and benefits from Sverdrup Technology, Inc. He graduated with a Doctor of Philosophy in Aerospace Engineering in May 2002.

Mr. Sebourn is a member of the American Institute of Aeronautics and Astronautics. He is a member of the Westwood Church of Christ in Tullahoma. He and his wife, Teresa Wilmoth Sebourn of Tullahoma, Tennessee, have one daughter, Lyndsey Ann Sebourn, and one son, Jared Lynn Dominic Sebourn.

3238 2553 49

07-24-02

MAB

