

To the Graduate Council:

I am submitting herewith a dissertation written by W. David Pointer entitled "Experimental Characterization of Flow Patterns and Flow Characteristics in the Bulk Mercury Flow Field of the Spallation Neutron Source Mercury Target." I have examined the final 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 Nuclear Engineering.

Dr. Arthur E. Ruggles

Arthur E. Ruggles, Major Professor

We have read this dissertation
and recommend its acceptance:

Dr. Masood Parang

Dr. Thomas. E. Shannon

Dr. Lawrence .W. Townsend

Accepted for the Council:

Dr. Anne Mayhew

Interim Vice Provost and
Dean of The Graduate School

(Original signatures are on file in the Graduate Records Office.)

Experimental Characterization of Flow Patterns and Flow Stability in the Bulk Mercury Flow Field of the Spallation Neutron Source Mercury Target

A dissertation presented for the completion of
the degree of Doctor of Philosophy

The University of Tennessee, Knoxville

W. David Pointer

May 2001

***“As far as the laws of mathematics
refer to reality, they are not certain;
and as far as they are certain,
they do not refer to reality.”***

-- Albert Einstein

Acknowledgements

The studies presented herein represent a milestone objective of the U.S. Department of Energy's Spallation Neutron Source project. Thank you to the SNS program office and the Oak Ridge National Laboratory for allowing me to join their team and for supporting this research. I would like to thank John Haines, Graydon Yoder, Jr., David Felde, Philip Jallouk, Darryl Cox, and George Farquharson from Oak Ridge National Laboratory for their invaluable guidance in developing test plans, advice on data collection techniques, and assistance in procurement of needed equipment. Special thanks also goes to Mark Wendel of Oak Ridge National Laboratory, for providing insight and advice in the development of these studies and allowing me to include a discussion of his computational fluid dynamics studies of the SNS mercury target in this dissertation.

I would like to express my gratitude to my advisor, Art Ruggles, for going well beyond the call of duty in his assistance with the development of these studies and this dissertation. I would also like to offer special thanks to Jason Crye for helping with all of those test section replacements and the low-tech process water replacement strategy when he had plenty of work to do in the heat transfer experiments. I would especially like to thank Patrick McClanahan for lending his invaluable expertise in the field of signal processing as well as dragging me out of the lab on a regular basis.

I owe my parents, Bill and Sue Pointer, a great debt of gratitude for their loving support, encouragement, and advice. I can't thank them enough for allowing me to always follow my dreams, even if they weren't always on the beaten path. Thanks to my sister, Suzanne, and brother-in-law, Bob, for providing much needed laughter and for being so supportive. Finally, thank you to my wife, Angela, for being kind and patient when things weren't going well, for being truly overjoyed when things went well, for lending her professional talents to the creation and editing of this document, and for not being afraid to embark on what is sure to be a life of endless adventure.

"Things derive their being and nature by mutual dependence and are nothing in themselves."

– Nagarjuna, Buddhist Scholar

Abstract

The Spallation Neutron Source will provide an intense pulsed source of neutrons for neutron scattering research by focusing a high-energy pulsed proton beam on a liquid mercury target. Interactions between the protons and the nuclei of the mercury will result in the production of neutrons through a spallation reaction. The use of liquid mercury obviates concerns with radiation damage associated with solid targets, facilitates removal of the heat deposited in the target, and allows online processing of the target material to reduce the concentration of the products of the spallation reaction. These advantages allow the use of a much higher energy proton beam than in existing facilities with solid targets.

While the use of liquid mercury has many advantages, the liquid mercury that flows through the target must provide sufficient heat transfer to maintain the temperature of the target structure within the thermal limits of the structural materials. Therefore, the liquid mercury flow field must be adequately characterized to provide an accurate evaluation of heat transfer in the SNS target. Since liquid mercury is completely opaque and corrosive to many materials, the use of liquid mercury as the working fluid makes characterization of the flow field by experiment difficult. Furthermore, the appearance of flow asymmetries and pseudo-periodic instabilities in the flow field is difficult to capture in Computational Fluid Dynamics models of the system using current technology. Thus, a thorough experimental program using well-scaled experiments is required to validate and tune the computational model for the evaluation of the SNS mercury target design.

The Spallation Neutron Source experimental program uses two scaled experiments to evaluate the fluid dynamic behavior of the bulk mercury flow of the SNS mercury target. The first facility uses air as a surrogate fluid for the liquid mercury, and the second facility uses water as a surrogate fluid. Flow visualization studies provide a qualitative evaluation of the flow behavior. Velocity mapping through Laser Doppler Velocimetry and Ultrasonic Doppler Profiling provide a more quantitative measure of the behavior of the flow field. The evaluation of the pressure field in the test sections provides insight into the nature of instabilities in the flow field. The evaluation of diffusion and dissipation in the flow field provides insight into the possible fluctuations in the temperature distribution in the mercury target. In general the experimental program indicates that the computational model provides a conservative evaluation of the flow behavior in the SNS target.

Table of Contents

Chapter 1 Introduction	1
1.1 Background	1
1.2 Design of the Spallation Neutron Source Target	3
1.3 Engineering Constraints	6
1.4 Purpose of the Experimental Program	7
Chapter 2 Single-Phase Turbulent Flow	8
2.1 The Equations of Motion for Turbulent Flow	8
2.2 Approximate Forms of the Equations of Motion	8
2.3 The Reynolds Equations of Turbulent Motion	10
2.4 Turbulence Modeling	15
Chapter 3 Description of Experimental Facilities	19
3.1 University of Tennessee Air Test Facility	19
3.2 The Water Thermal Hydraulic Loop	20
Chapter 4 Similarity and Scaling Studies Supporting the Experimental Program	27
4.1 Similarity of Planar Forces and Stresses	27
4.2 Similarity of Body Forces	40
4.3 Similarity of Forces Resulting from Energy Deposition	50
Chapter 5 Simulation of the SNS Target Bulk Mercury Flow	58
5.1 Description of the CFD Model	58
5.2 Predicted Flow Characteristics	60
5.3 Review of Previous Studies	62
Chapter 6 Flow Visualization Experiments to Characterize Flow Patterns in the SNS Mercury Target	65
6.1 Flow Visualization in the UTATF	65
6.2 Flow Visualization Studies in the WTHL Nominal Test Section	68

Chapter 7 Evaluation of Pressure Distributions and Flow Stability in the SNS Thermal Hydraulic Experiments	71
7.1 Evaluation of Inlet Flow Stability in the WTHL Experiments	71
7.2 Evaluation of Periodicity of Pressure Oscillations in the UTATF	73
7.3 Evaluation of Pressure Oscillations in the WTHL	75
Chapter 8 Evaluation of Time-Averaged Characteristics of the Velocity Field in the SNS Thermal Hydraulic Experiments	77
8.1 Mapping the Time-Averaged Velocity Field in the UTATF	77
8.2 Mapping the Time-Averaged Velocity Field in the WTHL Using Laser Doppler Velocimetry	77
8.3 Additional Velocity Measurements in the WTHL Nominal Test Section Using Ultrasonic Velocity Profiling	95
Chapter 9 Evaluation of Potential Flow Control Devices in the WTHL	105
9.1 Flow Visualization Studies in the Modified WTHL Test Section	106
9.2 Evaluation of the Pressure Field in the Modified WTHL Test Section	107
9.3 Evaluation of Time-Averaged Characteristics of the Velocity Field in the Modified WTHL Test Section	108
9.4 Repeatability of Velocity Measurements in the Modified WTHL Test Section	114
9.5 Comparison of Velocity Mappings in Nominal and Modified WTHL Test Sections	116
Chapter 10 Evaluation of Time-Dependent Characteristics of the Velocity Field in the SNS Thermal Hydraulic Experiments	118
10.1 Evaluation of Dissipation and Diffusion in the UTATF	118

10.2	Evaluation of Dissipation and Diffusion in the WTHL	133
Chapter 11 Evaluation of Structural Temperature		140
11.1	Evaluation of Structural Temperature Distributions from CFD models	140
11.2	Evaluation of Time-Dependent Fluctuation in the Baffle Wall Temperature	141
Chapter 12 Qualifications of the Ultrasonic Doppler Profiliometer		149
Chapter 13 Validation of the CFD Simulation of the SNS Mercury Target		151
13.1	Comparison of CFD Simulation to Flow Visualization Studies in the SNS Thermal Hydraulic Experiments	151
13.2	Comparison of Velocity Fields Predicted by the CFD Simulation and Measured in the WTHL	153
13.3	Conservative Versus Non-Conservative	158
Chapter 14 Conclusion		160
Chapter 15 Recommendations for Future Work		162
References		163
Appendix – Additional Velocity Maps from WTHL LDV Measurements		166
Vita		185

Table of Figures

Figure 1.1 Site plan of the Spallation Neutron Source	2
Figure 1.2 Mercury target vessel.	4
Figure 1.3 Cross-sectional view of the beam window region of the SNS target module	5
Figure 3.1 The University of Tennessee Air Test Facility.	19
Figure 3.2 Test section of the University of Tennessee Air Test Facility.	20
Figure 3.3 Schematic of the WTHL.	21
Figure 3.4 Sample Screen from WTHL Data Collection System	22
Figure 3.5 Water Thermal Hydraulic Loop test section.	22
Figure 3.6 Schematic of the WTHL test section.	23
Figure 3.7 Schematic of the WTHL test section and spool piece.	23
Figure 3.8 Two-beam Laser Doppler Velocimeter equipment.	24
Figure 3.9 WTHL test section with front and rear Plexiglas windows for LDV measurements.	25
Figure 3.10 Close-up view of the WTHL test section with single Plexiglas window.	25
Figure 4.1 Reynolds number in the SNS mercury target and the WTHL.	30
Figure 4.2 Coriolis acceleration free body diagram	44
Figure 4.3 Development of Taylor vortices in an annular region between rotating cylinders.	48
Figure 4.4 Development of Goertler vortices along a concave surface.	49
Figure 4.5 Closed pipe geometry used for bounding evaluation of thermal shock effects.	55
Figure 5.1 Target geometry included in the three-dimensional CFD model.	58
Figure 5.2 Close-up view of the discretization of the SNS mercury target.	59

Figure 5.3 Streamlines in the SNS mercury target with the assumption of horizontal symmetry.	60
Figure 5.4 Velocity profiles from the steady state two-dimensional simulations of the SNS mercury target.	61
Figure 5.5 Simulation of the SNS target flow assuming vertical symmetry.	61
Figure 6.1 Flow visualization in the UTATF using yarn tufts.	65
Figure 6.2 Flow visualization experiment in the UTATF at half nominal flow.	66
Figure 6.3 Flow visualziation experiment in the UTATF at nominal flow.	66
Figure 6.4 Thermochromic imaging in the UTATF.	67
Figure 6.5 Three characteristic modes of flow behavior in te UTATF as visualized by thermochromic imaging.	67
Figure 6.6 Flow visualization in the WTHL test section at nominal flow using dye injection.	68
Figure 6.7 Flow visualization in the WTHL test section at half nominal flow using dye iujection.	69
Figure 6.8 Flow visualization in the WTHL test section at nominal flow using bubble injection.	70
Figure 6.9 Flow visualization in teh WTHL test section at half-nominal flow using bubble injection.	70
Figure 7.1 Pressure instrumentation used in the evaluation of inlet flow stability in the WTHL.	71
Figure 7.2 Pressure drop ratio versus Reynolds number.	72
Figure 7.3 Pressure drop ratio versus the dynamic head.	73
Figure 7.4 Variations in local pressure in the UTATF at a position located on the centerline of the exit channel that is 1 inch upstream of the leading edge of the flow baffles.	73
Figure 7.5 Variations in local pressure in the UTATF at a position located on the centerline of the exit channel that is 3 inches downstream of the leading edge of the flow baffles.	74

Figure 7.6 Short-term Fourier transform of the pressure signal shown in Figure 7.4	74
Figure 7.7 Short-term Fourier transform of the pressure signal shown in Figure 7.5	75
Figure 7.8 Pressure tap locations for evaluation of test section pressure losses.	76
Figure 8.1 Time-averaged total velocity magnitude in the UTATF test section at 1.5 inches above the centerline.	78
Figure 8.2 Time-averaged total velocity magnitude in the UTATF test section at the centerline.	79
Figure 8.3 Time-averaged total velocity magnitude in the UTATF test section at 1.5 inches below the centerline.	80
Figure 8.4 Velocity in front window of the nominal test section at a depth of 10 mm from the upper inside surface.	82
Figure 8.5 Turbulence intensity in front window of the nominal test section at a depth of 10 mm from the upper inside surface.	83
Figure 8.6 Velocity in front window of the nominal test section at a depth of 20 mm from the upper inside surface.	83
Figure 8.7 Turbulence intensity in front window of the nominal test section at a depth of 20 mm from the upper inside surface.	84
Figure 8.8 Velocity in front window of the nominal test section at a the horizontal midplane.	84
Figure 8.9 Turbulence intensity in front window of the nominal test section at the horizontal midplane.	85
Figure 8.10 Velocity in front window of the nominal test section at a depth of 20 mm from the lower inside surface.	85
Figure 8.11 Turbulence intensity in front window of the nominal test section at a depth of 20 mm from the lower inside surface.	86
Figure 8.12 Velocity in front window of the nominal test section at a depth of 10 mm from the lower inside surface.	86
Figure 8.13 Turbulence intensity in front window of the nominal test section at a depth of 10 mm from the lower inside surface.	87

Figure 8.14 Velocity in nominal test section at a depth of 10 mm from the upper inside surface.	87
Figure 8.15 Turbulence intensity in the nominal test section at a depth of 10 mm from the upper inside surface.	88
Figure 8.16 Velocity in nominal test section at a depth of 20 mm from the upper inside surface.	88
Figure 8.17 Turbulence intensity in the nominal test section at a depth of 20 mm from the upper inside surface.	89
Figure 8.18 Velocity in nominal test section at horizontal midplane.	89
Figure 8.19 Turbulence intensity in the nominal test section at horizontal midplane.	90
Figure 8.20 Velocity in nominal test section at a depth of 20 mm from the lower inside surface.	90
Figure 8.21 Turbulence intensity in the nominal test section at a depth of 20 mm from the lower inside surface.	91
Figure 8.22 Velocity in nominal test section at a depth of 10 mm from the lower inside surface.	91
Figure 8.23 Turbulence intensity in the nominal test section at a depth of 10 mm from the lower inside surface.	92
Figure 8.24 Comparison of velocity measurements at horizontal midplane and a distance of 70.5 mm from the test section node in nominal test section.	93
Figure 8.25 Comparison of velocity measurements at horizontal midplane and a distance of 1100.5 mm from the test section node in nominal test section.	93
Figure 8.26 Comparison of velocity measurements at horizontal midplane and a distance of 150.5 mm from the test section node in nominal test section.	94
Figure 8.27 Comparison of velocity measurements at horizontal midplane and a distance of 190.5 mm from the test section node in nominal test section.	94

Figure 8.28 Comparison of velocity measurements at horizontal midplane and a distance of 110.5 mm from the test section node in nominal test section., with a flow rate of 5.5 L/s per inlet leg.	96
Figure 8.29 Comparison of velocity measurements at horizontal midplane and a distance of 150.5 mm from the test section node in nominal test section., with a flow rate of 5.5 L/s per inlet leg.	96
Figure 8.30 Comparison of velocity measurements at horizontal midplane and a distance of 190.5 mm from the test section node in nominal test section., with a flow rate of 5.5 L/s per inlet leg.	97
Figure 8.31 Comparison of velocity measurements at horizontal midplane and a distance of 70.5 mm from the test section node in nominal test section., with a flow rate of 7.7 L/s per inlet leg.	97
Figure 8.32 Comparison of velocity measurements at horizontal midplane and a distance of 110.5 mm from the test section node in nominal test section., with a flow rate of 7.7 L/s per inlet leg.	98
Figure 8.33 Comparison of velocity measurements at horizontal midplane and a distance of 150.5 mm from the test section node in nominal test section., with a flow rate of 7.7 L/s per inlet leg.	98
Figure 8.34 Comparison of velocity measurements at horizontal midplane and a distance of 190.5 mm from the test section node in nominal test section., with a flow rate of 7.7 L/s per inlet leg.	99
Figure 8.35 Schematic showing typical position of the UVP probe.	99
Figure 8.36 Time-averaged velocity away from the test section nose at a distance of 4 inches to the right of the centerline.	100
Figure 8.37 Time-averaged velocity away from the test section nose at a distance of 3 inches to the right of the centerline.	101
Figure 8.38 Time-averaged velocity away from the test section nose at a distance of 2 inches to the right of the centerline.	101
Figure 8.39 Time-averaged velocity away from the test section nose at a distance of 1 inch to the right of the centerline.	102
Figure 8.40 Time-averaged velocity away from the test section nose at the centerline of the exit channel.	103

Figure 8.41 Time-averaged velocity away from the test section nose at a distance of 1 inch to the left of the centerline.	103
Figure 8.42 Time-averaged velocity away from the test section nose at a distance of 2 inches to the left of the centerline.	103
Figure 8.43 Time-averaged velocity away from the test section nose at a distance of 3 inches to the left of the centerline.	104
Figure 8.44 Time-averaged velocity away from the test section nose at a distance of 4 inches to the left of the centerline.	104
Figure 9.1 Close-up view of region surrounding the leading edge of the flow baffle from results of a CFD simulation of the flow field in the modified WTHL test section.	105
Figure 9.2 Flow visualization experiment in the modified WTHL test section with dye injected along the inside surface of the flow vane.	106
Figure 9.3 Flow visualization experiment in the modified WTHL test section with dye injected along the outside surface of the flow vane.	106
Figure 9.4 Velocity in modified test section at a depth of 10 mm from the upper inside surface.	109
Figure 9.5 Velocity in modified test section at a depth of 20 mm from the upper inside surface.	109
Figure 9.6 Velocity in modified test section at the horizontal midplane.	110
Figure 9.7 Velocity in modified test section at a depth of 20 mm from the lower inside surface.	110
Figure 9.8 Velocity in modified test section at a depth of 10 mm from the lower inside surface.	111
Figure 9.9 Turbulence intensity in modified test section at a depth of 10 mm from the upper inside surface.	111
Figure 9.10 Turbulence intensity in modified test section at a depth of 20 mm from the upper inside surface.	112
Figure 9.11 Turbulence intensity in modified test section at the horizontal midplane.	112

Figure 9.12 Turbulence intensity in modified test section at a depth of 20 mm from the lower inside surface.	113
Figure 9.13 turbulence intensity in modified test section at a depth of 10 mm from the lower inside surface.	113
Figure 9.14 Velocity comparison at a distance of 110.5 mm form test section nose.	114
Figure 9.15 Velocity comparison at a distance of 150.5 mm form test section nose.	115
Figure 9.16 Velocity comparison at a distance of 190.5 mm form test section nose.	115
Figure 9.17 Comparison of the LDV velocity data in the nominal and modified WTHL test section.	116
Figure 9.18 Comparison of LDV turbulence intensity data in the nominal and modified WTHL test section.	117
Figure 10.1 Voltage signal from high speed thermocouple located at the test section centerline.	119
Figure 10.2 Voltage signal from high speed thermocouple located 2 inches from the test section centerline.	120
Figure 10.3 Voltage signal from high speed thermocouple located 4 inches from the test section centerline.	120
Figure 10.4 Voltage signal from high speed thermocouple located 6 inches from the test section centerline.	121
Figure 10.5 Voltage signal from high speed thermocouple located 7.5 inches from the test section centerline.	121
Figure 10.6 Voltage signal from the control thermocouple located in a closed container	122
Figure 10.7 Short term Fourier transform of voltatge signal from high-speed thermocouple located at the test section centerline.	123
Figure 10.8 Short term Fourier transform of voltage signal from high-speed thermocouple located 2 inches from the test section centerline.	123

Figure 10.9 Short term Fourier transform of voltage signal from high-speed thermocouple located 4 inches from the test section centerline.	124
Figure 10.10 Short term Fourier transform of voltage signal from high-speed thermocouple located 6 inches from the test section centerline.	124
Figure 10.11 Short term Fourier transform of voltage signal from high-speed thermocouple located 7.5 inches from the test section centerline.	125
Figure 10.12 Short term Fourier transform of voltage signal from the control thermocouple in a closed container.	125
Figure 10.13 Morlet wavelet used in the evaluation of voltage signals from the high-speed thermocouples used in the UTATF diffusion studies.	126
Figure 10.14 Continuous 1-D wavelet transform of voltage signal from high-speed thermocouple located at the test section centerline.	127
Figure 10.15 Continuous 1-D wavelet transform of voltage signal from high-speed thermocouple located 2 inches from the test section centerline.	128
Figure 10.16 Continuous 1-D wavelet transform of voltage signal from high-speed thermocouple located 4 inches from the test section centerline.	129
Figure 10.17 Continuous 1-D wavelet transform of voltage signal from high-speed thermocouple located 6 inches from the test section centerline.	130
Figure 10.18 Continuous 1-D wavelet transform of voltage signal from high-speed thermocouple located 7.5 inches from the test section centerline.	131
Figure 10.19 Continuous 1-D wavelet transform of voltage signal from control thermocouple located in a closed container.	132

Figure 10.20 Flow visualization experiment in the WTHL for evaluation of fluid residence times in the primary recirculation zones.	133
Figure 10.21 Time-dependent velocity away from the test section nose at a distance of 4 inches to the left of the centerline of the exit channel.	135
Figure 10.22 Time-dependent velocity away from the test section nose at a distance of 3 inches to the left of the centerline of the exit channel.	135
Figure 10.23 Time-dependent velocity away from the test section nose at a distance of 2 inches to the left of the centerline of the exit channel.	136
Figure 10.24 Time-dependent velocity away from the test section nose at a distance of 1 inch to the left of the centerline of the exit channel.	136
Figure 10.25 Time-dependent velocity away from the test section nose at the centerline of the exit channel.	137
Figure 10.26 Time-dependent velocity away from the test section nose at a distance of 1 inch to the right of the centerline of the exit channel.	137
Figure 10.27 Time-dependent velocity away from the test section nose at a distance of 2 inches to the right of the centerline of the exit channel.	138
Figure 10.28 Time-dependent velocity away from the test section nose at a distance of 3 inches to the right of the centerline of the exit channel.	138
Figure 10.29 Time-dependent velocity away from the test section nose at a distance of 4 inches to the right of the centerline of the exit channel.	139
Figure 11.1 Temperature distribution in the bulk mercury flow and on the surface of the flow baffle within the Spallation Neutron Source mercury target.	141

Figure 11.2 Local Variation in bulk fluid temperature as a function of variation in fluid residence time.	144
Figure 11.3 Local variation in the heat transfer coefficient as a function of the magnitude of the variation in the flow velocity.	146
Figure 11.4 Variation in the surface temperature of the flow baffle as a result of variation in the bulk fluid temperature.	146
Figure 11.5 Variation in the surface temperature of the flow baffle as a result of variation in the flow velocity.	147
Figure 11.6 Variation in the surface temperature of the flow baffle as a result of variations in the flow velocity and bulk fluid temperature.	148
Figure 12.1 Comparison of LDV measurements and selected UVP measurements in the nominal WTHL test section.	149
Figure 12.2 Comparison of LDV measurements and all UVP measurements in the nominal WTHL test section.	150
Figure 13.1 Streamlines predicted by the CFD simulation of the nominal WTHL test section at half nominal flow.	152
Figure 13.2 Flow visualization experiment using bubble injection in the nominal WTHL test section at half nominal flow.	152
Figure 13.3 Comparison of the LDV velocity measurements in the WTHL test section at nominal flow with predicted values from the CFD model.	154
Figure 13.4 Comparison of the axial component of the turbulence intensity from LDV measurements in the nominal WTHL test section at the nominal flow velocity and CFD simulation.	155
Figure 13.4 Comparison of the cross-stream component of the turbulence intensity from LDV measurements in the nominal WTHL test section at the nominal flow velocity and CFD simulation.	156
Figure 13.6 Comparison of multiple LDV velocity data sets and predicted velocity values.	157
Figure 13.7 Comparison of multiple LDV cross-stream turbulence intensity data sets and predicted values.	157

Figure 13.8 Close-up view of predicted and measured velocity vectors in the recirculation zone region near the leading edge of the flow baffle.	158
Figure A.1 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm form the upper inside surface of the test section.	163
Figure A.2 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm form the upper inside surface of the test section.	163
Figure A.3 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm form the upper inside surface of the test section.	164
Figure A.4 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm form the upper inside surface of the test section.	164
Figure A.5 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm form the upper inside surface of the test section.	165
Figure A.6 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm form the upper inside surface of the test section.	165
Figure A.7 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 20 mm form the upper inside surface of the test section.	166
Figure A.8 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 20 mm form the upper inside surface of the test section.	166
Figure A.9 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm form the upper inside surface of the test section.	167

Figure A.10 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 20 mm from the upper inside surface of the test section.	167
Figure A.11 Velocity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.	168
Figure A.12 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.	168
Figure A.13 Velocity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.	169
Figure A.14 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.	169
Figure A.15 Velocity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.	170
Figure A.16 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.	170
Figure A.17 Velocity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.	171
Figure A.18 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.	171
Figure A.19 Velocity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.	172
Figure A.20 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.	172
Figure A.21 Velocity measured in the nominal WTHL test section at 70 percent of the nominal flow rate at the test section centerline.	173
Figure A.22 Turbulence intensity measured in the nominal WTHL test section at 70 percent of the nominal flow rate at the test section centerline.	173
Figure A.23 Velocity measured in the nominal WTHL test section at 70 percent of the nominal flow rate at the test section centerline.	174

Figure A.24 Turbulence intensity measured in the nominal WTHL test section at 70 percent of the nominal flow rate at the test section centerline.	174
Figure A.25 Velocity measured in the nominal WTHL test section at 50 percent of the nominal flow rate at the test section centerline.	175
Figure A.26 Turbulence intensity measured in the nominal WTHL test section at 50 percent of the nominal flow rate at the test section centerline.	175
Figure A.27 Velocity measured in the nominal WTHL test section at 50 percent of the nominal flow rate at the test section centerline.	176
Figure A.28 Turbulence intensity measured in the nominal WTHL test section at 50 percent of the nominal flow rate at the test section centerline.	176
Figure A.29 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 20 mm from the bottom inside surface of the test section.	177
Figure A.30 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 20 mm from the bottom inside surface of the test section.	177
Figure A.31 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section.	178
Figure A.32 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section.	178
Figure A.33 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section.	179
Figure A.33 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section.	179

Figure A.35 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section.	180
Figure A.36 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section.	180

Nomenclature

Variables

$\vec{A}_c$	Coriolis acceleration
E	Internal energy
E	Ekman number
g	magnitude of gravitational acceleration
$\vec{g}$	gravitational vector
Go	Goertler Number
Gr	Grashof Number
$\vec{G}$	force field vector
H	enthalpy
k	thermal conductivity
$\hat{k}$	unit vector in z-direction
K	turbulent kinetic energy
l_m	Prandtl mixing length
L	characteristic length
p	pressure
q	turbulent fluctuation in the magnitude of the velocity
$\vec{r}$	position vector
r, R	radius
r_c	radius of curvature
Re	Reynolds number
Ri	Richardson number
t	time
T	temperature
Ta	Taylor number
Ti	turbulence intensity
u	first component of the velocity vector

u'	turbulent fluctuation in the first component of the velocity vector
u_i	i th component of the velocity vector
U	magnitude of the flow velocity
v	second component of the velocity vector
v'	turbulent fluctuation in the second component of the velocity vector
$\vec{v}$	velocity vector
w	third component of the velocity vector
w'	turbulent fluctuation in the third component of the velocity vector
x	first component of the position vector
x_i	i th component of the position vector
y	second component of the position vector
y_+	length scale
z	third component of the position vector
β	volumetric thermal expansion
δ	boudary layer thickness
δ_{ij}	Kroenecker delta function
ε	eddy diffusivity
ϕ	scalar potential
μ	dynamic viscosity
ν	kinematic viscosity
θ	temperature defect
ρ	density
σ	shear stress magnitude
τ	shear stress component
τ_{ij}	shear stress tensor
Ω	magnitude of the angular velocity
$\vec{\Omega}$	angular velocity vector

$\vec{\nabla}$

gradient, or vector derivative

Subscripts

i

Indicial parameter

j

Indicial parameter

i

corresponding to inner radius

k

Indicial parameter

m

momentum

o

corresponding to outer radius

r

corresponding to radius

ref

reference value

s

property of the surface

t

turbulent

w

property of the wall

x

component corresponding to the x-direction

y

component corresponding to the y-direction

z

component corresponding to the z-direction

0

Initial or reference value

ε

corresponding to turbulent diffusion

∞

property of the bulk fluid flow

Chapter 1 Introduction

With any complex engineering system, the most challenging part of design and development is often the verification that the proposed design satisfies the desired safety and operational constraints. Design verification is particularly challenging when the system includes hazardous or corrosive materials. When this is the case, computer simulation is often the only means of evaluating the viability of the proposed design. However, the abilities of simulation software are quite limited, especially for turbulent fluid systems. Therefore, it is often necessary to verify the abilities and applicability of the Computational Fluid Dynamics (CFD) software before it can be used to evaluate the design. The validation of the software requires data from carefully designed experiments that replicate the expected behavior of the fluid within the proposed system. The purpose of this study is to provide a high fidelity validation of the computational model used to evaluate the viability of the design of the liquid mercury target of the Spallation Neutron Source (SNS).

1.1 Background

Neutron scattering research provides valuable information on the structure of materials that cannot be obtained using other methods. Neutron scattering can be used to determine the arrangement, motion, and interaction of atoms within a material. With a better understanding of the material structure, improvements can be made to enhance material performance in specific applications. Ultimately, neutron scattering research results in an improvement in the quality of consumer products.

Pulsed Neutron Sources

While many neutron sources are available worldwide for scientific research, demand has outgrown capacity, and more intense sources are needed for more advanced scattering research. On the recommendation of the Basic Energy Sciences Advisory Committee (BESAC) on Neutron Science for America's Future, the U.S. Department of Energy began to pursue development of a new pulsed neutron source in 1996.

Neutron scattering research benefits from having a variety of neutron sources available, including both continuous sources, such as reactors, and pulsed sources, such as accelerator-based sources. In many cases, pulsed sources provide

more information about the material of interest than can be obtained using continuous sources. Furthermore, advances in accelerator technology allow the development of pulsed neutron sources with intensities much greater than those found in the brightest continuous sources. Intense short-pulse neutron beams allow researchers to analyze the motion of both the incident neutrons and the atoms within the material to obtain information that cannot be evaluated using continuous sources. Such a neutron source will provide opportunities to practically every scientific and technical field of study.

Accelerator-based pulsed neutron sources produce neutrons through spallation of a heavy metal target. When a high-energy particle, such as a proton, collides with a heavy atomic nucleus, some of the neutrons are immediately knocked out of the nucleus or “spalled.” The collision also increases the energy of the particles within the nucleus. As a result, additional neutrons may have sufficient energy to escape the nuclear forces that bind them in the nucleus and “boil off.” For every proton striking a nucleus, 20 to 30 free neutrons may be produced (Spallation Neutron Source: Advancing the Frontiers of Science and Technology, 2000).

The Spallation Neutron Source

Through a partnership of six Department of Energy national laboratories (Argonne, Brookhaven, Jefferson, Lawrence Berkeley, Los Alamos, and Oak Ridge) the SNS project was developed, and a conceptual design was completed in May 1997. With additional technical, cost and schedule developments by the partnership, construction began in December 1999. The SNS will be located on the Oak Ridge Reservation between the Oak Ridge National Laboratory and the Y-12 Weapons Complex. An overview of the SNS site is shown in Figure 1.1.

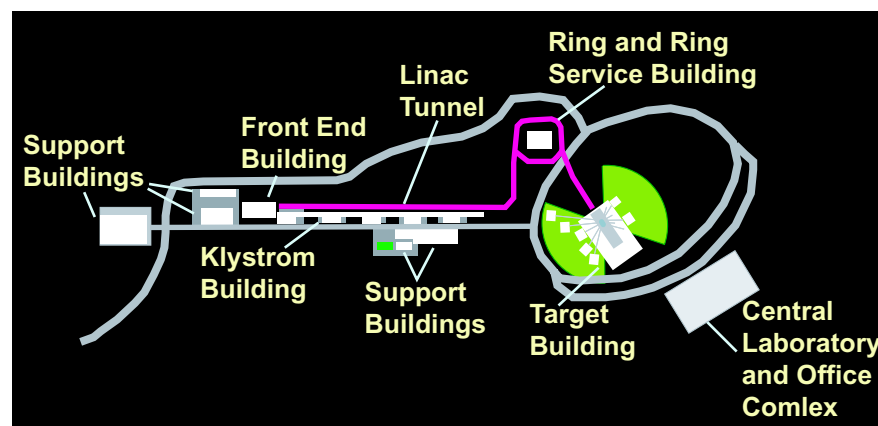


Figure 1.1 – Site plan of the Spallation Neutron Source.

Neutrons are produced in the SNS by a high-energy proton beam incident on a liquid mercury target. An ion source produces negatively charged hydrogen ions that are injected into a linear accelerator with an energy of 2.5 million electron volts (2.5 MeV). The ions are accelerated to very high energies (approximately 1 GeV) and sent through a foil to strip off the two electrons orbiting the single proton in the ion's nucleus. The proton beam is then passed into an accumulator ring where it is compressed into short pulses. The high-energy proton pulse is released from the accumulator ring and impinges on the liquid mercury target. A pulse of neutrons is freed from the mercury by the proton pulse. The energy spectrum of neutron pulse is modified as it passes through a moderator, and the pulse travels through beam lines to the various experiment stations (Spallation Neutron Source: Advancing the Frontiers of Science and Technology, 2000).

The SNS will be the first scientific facility to use a liquid mercury target for the production of neutrons through spallation. Liquid mercury has several properties that make it a good choice for the SNS target material. Liquid mercury is not damaged by radiation as most solid materials are. Mercury has a high mass number, and many neutrons are produced by a single spallation reaction. Since mercury is a liquid, temperature and pressure waves induced by the proton pulses can be dissipated more easily than in a solid target (Spallation Neutron Source: Advancing the Frontiers of Science and Technology, 2000). However, the use of a flowing liquid target requires a thorough understanding of the dynamic behavior of the flow as it moves through the target to ensure that the relevant temperature and material stress limits are not exceeded during system operation.

1.2 Design of the Spallation Neutron Source Target

The SNS mercury target must satisfy three major performance requirements. First, the target must provide sufficient quantities of neutrons through spallation of the target material by the high-energy proton beam. Second, the target structure must safely contain the liquid mercury target material. Finally, the target system must be capable of transporting the proton beam power and the resulting nuclear decay power away from the region of the target struck by the proton beam and to secondary cooling systems.

For the first performance requirement to be satisfied, a sufficient number of protons must strike the nuclei of the liquid mercury atoms. Therefore, the target must be sufficiently large, so that very few of the incident protons escape the target

without interacting with the mercury nuclei. Furthermore, the structural and cooling materials in the beam window region of the target should be minimized to ensure that the maximum number of protons reach the liquid mercury.

The second and third performance requirements indicate that the target must be capable of removing the heat and withstanding the pulsed effects of a 1 GeV proton beam with a time-averaged power of approximately 2 MW and a pulse frequency of approximately 60 Hz. Furthermore, all components of the target must be designed with at least two independent structures surrounding the liquid mercury to reduce the likelihood of releasing liquid mercury to the surrounding environments (Spallation Neutron Source Design Manual, 1998).

Nominal Target Design

The SNS target module consists of the mercury target vessel, the water-cooled shroud that surrounds the target vessel, and the mercury and water feed and return lines. The mercury target vessel is illustrated in Figure 1.2. In order to provide a localized containment in the event of an accident, the mercury target vessel is completely surrounded by a water-cooled shroud. Between the two structures is a

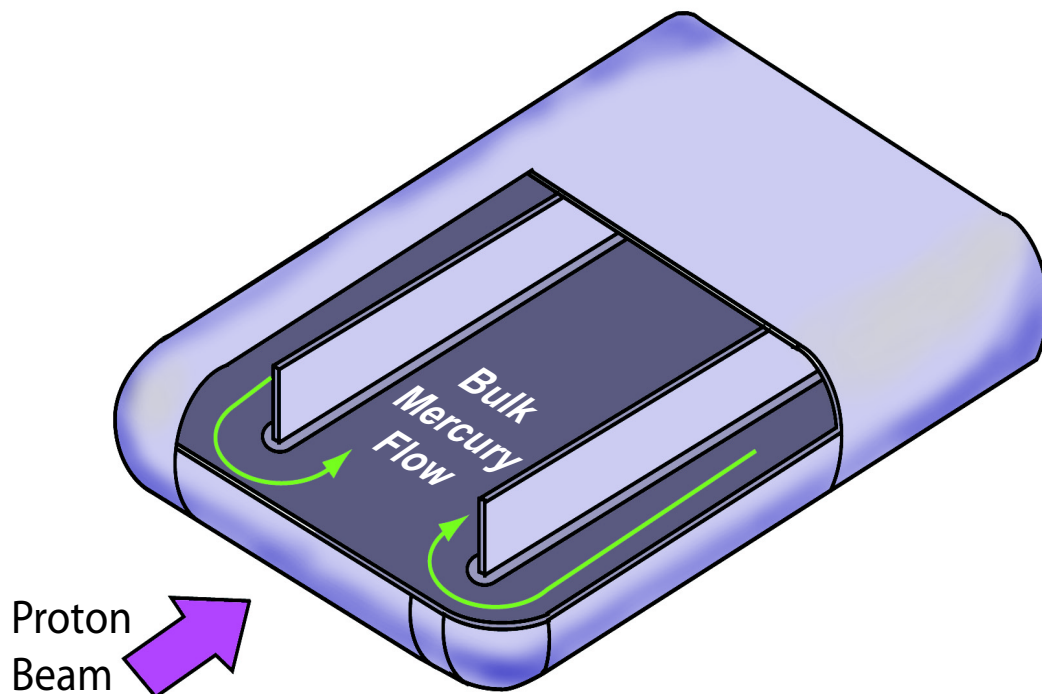


Figure 1.2 Mercury target vessel.

helium gap for the detection of mercury vapor should a leak develop in the mercury target vessel. A cross-sectional view of the beam window region of the target module is shown in Figure 1.3.

Mercury Target Vessel

The mercury target is contained in a 316-type stainless steel mercury target vessel. During operation, relatively low mercury temperatures ($<200\text{ }^{\circ}\text{C}$) must be maintained to prevent dissolution of the nickel from the 316-type stainless steel alloy. The vessel, as illustrated in Figure 1.2, has a width of 40.4 cm, a height of 10.4 cm, and a length of 77.45 cm. The bulk mercury flow enters through the two side channels that are outside the region of the target that is visible to the proton beam. These channels have a radius of 8.5 cm and the flow rate through each inlet is 11 L/s. The two inlet flows exit the target through the common center channel that has a width of 22.4 cm and a height of 8.0 cm. The target vessel has a double wall structure, creating a separate mercury flow channel that cools the beam window region of the target. As a result, the cooling of the structure in

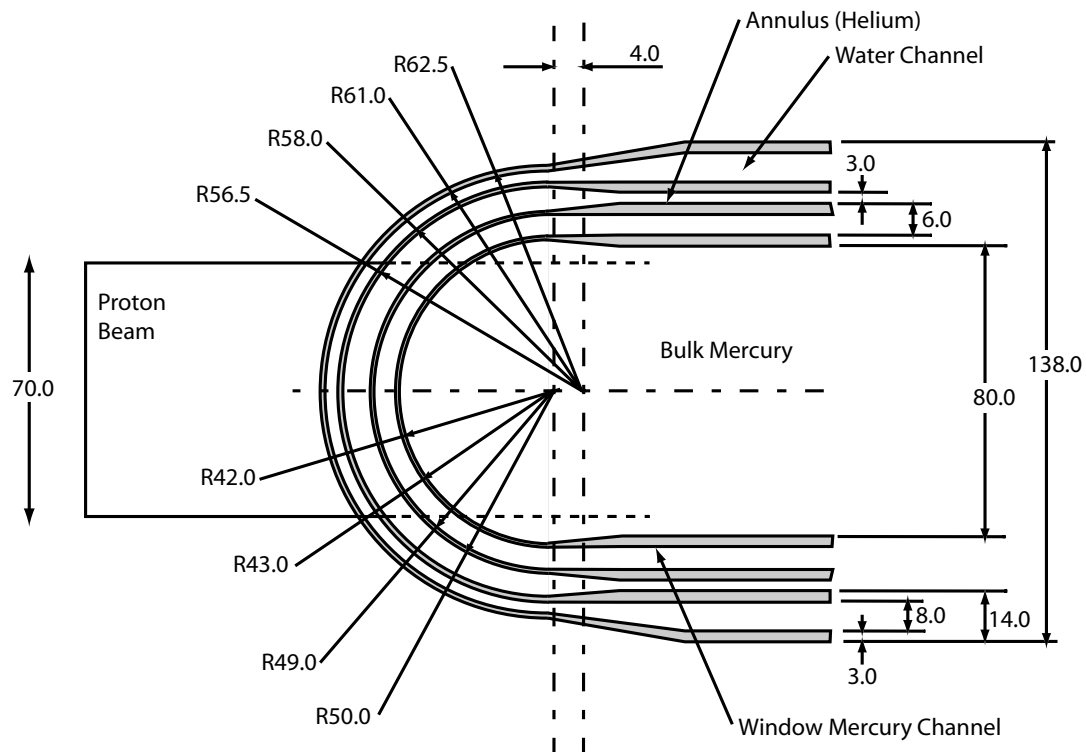


Figure 1.3 Cross-sectional view of the beam window region of the SNS target module.

the beam window region and the transport of heat deposited in the bulk mercury are achieved by separate flow paths (Spallation Neutron Source Design Manual, 1998).

Water-Cooled Shroud

The mercury target vessel is surrounded by a water-cooled shroud that serves as a mercury containment vessel in the event of a leak in the target vessel. The shroud guides any mercury that leaks from the target vessel to the mercury dump tank. The gap between the mercury target vessel and the water-cooled shroud is filled with helium and instrumented to detect mercury vapor that indicates the presence of a leak in the target vessel. Water cooling of the shroud is necessary to remove the heat deposited in the stainless steel structure by the proton beam (Spallation Neutron Source Design Manual, 1998).

1.3 Engineering Constraints

The primary design constraints for the development of the SNS mercury target relate to the characteristics of the neutrons that are produced in the mercury target. These constraints must be met in order to insure that the design meets the operational requirements of the facility. The target design is also subject to a number of engineering constraints that result from the properties of the materials used. These constraints must be satisfied to insure that the design meets the safety requirements of the facility.

The primary engineering constraints of concern in this study result from the thermal limits of the materials used in the system. The solid structure of the mercury target is constructed of stainless steel. During the operation of the SNS, this material is subjected to many stresses resulting from the energy deposited by the pulsed proton beam. In order to maximize the average lifetime of a target module, it is desirable to maintain the temperature of the target structure below 190 °C.

During operation, it is also desirable to maintain the mercury temperature below 210 °C in order to reduce the dissolution of nickel from the stainless steel and the likelihood of the growth and collapse of mercury vapor bubbles, or cavitation, within the flow field. The collapse of a cavitation bubble results in a large pressure disturbance adjacent to the solid structure where the bubble was formed, which can cause erosion of the structure, commonly known as cavitation damage. The combination of nickel dissolution and cavitation damage would reduce the ability of

the structural materials to withstand the stresses that result from normal operation and reduce the lifetime of the target.

1.4 Purpose of the Experimental Program

In order to ensure that the design of the SNS mercury target satisfies the safety, reliability, and availability requirements set forth by the SNS program, the thermal and mechanical stresses present in the structure of the mercury target must be well understood for conceivable operating conditions. The evaluation of thermal stresses in the target structure requires that the temperature distribution within the target structure be well known. Since the development and operation of an experiment to directly observe the temperature distribution would be technically difficult and prohibitively expensive, the analysis of the temperature distribution is limited to computational prediction using the CFD simulation code CFX. Scaled thermal hydraulic experiments will provide data to be compared with the simulation for validation of the computational results.

The purpose of the studies discussed herein is to verify that the predictions of the CFD simulation provide a conservative representation of the dynamic behavior of the fluid flow field within the SNS liquid mercury target. In order to provide a high fidelity verification of a computational fluid dynamics model, a high fidelity data set from an experiment that exhibits behavior similar to the behavior of the modeled flow field must be available. The creation of such a data set requires a well-designed experiment in which the similarity to the flow field of interest can be verified and the instrumentation in the facility can be relied upon to capture phenomena of interest.

Chapter 2 Single-Phase Turbulent Flow

The typical engineer's familiarity with turbulent flow often begins and ends with the knowledge of the existence of a critical velocity for flow through a circular pipe. The critical velocity provides the boundary between the regime of steady laminar flow and the region of irregular turbulent flow. In flows that exhibit the random turbulent behavior, the coefficients of viscosity, heat conduction and diffusion are effectively increased. It is well known that these net effects result from the development of a complex flow field consisting of interacting eddy structures which convect mass, momentum and energy from one region of the flow field to another.

2.1 The Equations of Motion for Turbulent Flow

Customarily, the turbulent motion of fluids is assumed to be described by the equations of motion for a continuous fluid without a molecular structure. The assumption is valid for any fluid in which the characteristic velocity of the characteristic turbulence structure, or eddy, does not exceed the characteristic molecular velocity for the fluid.

The equations of motion for a continuous fluid without a molecular structure are derived from the conservation laws and properties of the fluid. They consist of the equation of state, the relation between the stress and the strain rate, and the relation between the conducted heat flux and the temperature gradient. These equations are more commonly referred to as the conservation of mass, conservation of momentum, and conservation of energy equations for turbulent fluid flow.

The vector mass flux at any point in the flow field is given by the product of the density and the vector velocity at that point, $\rho \mathbf{u}_i$. The conservation of mass in the flow field is expressed mathematically by equating the divergence of the mass flux and the rate of change in the local density as shown in Equation 2.1. This

Equation 2.1

$$\frac{\partial u_i}{\partial x_i} + \frac{1}{\rho} \frac{D\rho}{Dt} = 0$$

formulation uses the usual summation convention where repeated free suffixes imply summation over three variables, shown in Equation 2.2. The formulation

Equation 2.2

$$\frac{\partial u_i}{\partial x_i} = \frac{\partial u_1}{\partial x_1} + \frac{\partial u_2}{\partial x_2} + \frac{\partial u_3}{\partial x_3}$$

also uses the material derivative, defined in Equation 2.3, which measures the rate of change following a fluid particle. In other words, the derivative considers the rate of change in a differential volume that moves with the fluid.

Equation 2.3

$$\frac{D}{Dt} \equiv \frac{\partial}{\partial t} + u_i \frac{\partial}{\partial x_i}$$

The momentum flux in the x_i -direction across a differential surface normal to the x_i -direction is given by the tensor $\rho u_i u_j$. The conservation of momentum is expressed by equating the difference between the rate of change of the momentum and the divergence of the momentum flux to the rate of change resulting from external forces, molecular forces, and inflow or outflow of momentum across the boundaries of the differential control volume. If the external forces, such as gravity, are described by a scalar potential function ϕ , the conservation of momentum equation can be expressed as shown in Equation 2.4. Using the conservation of

Equation 2.4

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} - \rho \frac{\partial \phi}{\partial x_i}$$

mass equation, Equation 2.1, the momentum equation can be rewritten using the material derivative as shown in Equation 2.5.

Equation 2.5

$$\rho \frac{Du_i}{Dt} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} - \rho \frac{\partial \phi}{\partial x_i}$$

The convected energy flux is given by Equation 2.6, where the term in parenthesis is the energy per unit mass of the fluid – the sum of the internal energy, the kinetic energy of the fluid motion, and the potential energy. The conservation of energy equation, shown in Equation 2.7, is derived by equating the sum of the rate of

Equation 2.6

$$\text{energy flux} = \rho u_i \left(E + \frac{1}{2} u_i^2 + \phi \right)$$

Equation 2.7

$$\begin{aligned} \frac{\partial}{\partial t} \left(\rho \left(E + \frac{1}{2} u_i^2 + \phi \right) \right) + \frac{\partial}{\partial x_i} \left[\rho u_i \left(E + \frac{1}{2} u_i^2 + \phi \right) \right] \\ = - \frac{\partial (p u_i)}{\partial x_i} + \frac{\partial (\tau_{ij} u_{ji})}{\partial x_i} + \frac{\partial}{\partial x_i} \left(k \frac{\partial T}{\partial x_i} \right) \end{aligned}$$

change in the energy of the fluid and the divergence of the energy flux to the rate of change resulting from stresses acting on the fluid and the conducted flux of thermal energy. An additional term representing the volumetric heat generation may be added to the energy equation but is neglected in this development. The energy equation can be simplified by recognizing that Equation 2.5 can be modified to obtain an equation for the kinetic energy of the fluid motion as shown in Equation 2.8. The simplified expression of the conservation of energy is shown in Equation 2.9. Typically, the energy equation is further simplified by defining the enthalpy as shown in Equation 2.10, and rewriting the expression as shown in Equation 2.11 (Townsend, 1976).

Equation 2.8

$$\rho \frac{D}{Dt} \left(\frac{1}{2} u_i^2 \right) = -u_i \frac{\partial p}{\partial x_i} + u_i \frac{\partial \tau_{ij}}{\partial x_j} - \rho u_i \frac{\partial \phi}{\partial x_i}$$

Equation 2.9

$$\rho \frac{DE}{Dt} = -p \frac{\partial u_j}{\partial x_j} + \tau_{ij} \frac{\partial u_i}{\partial x_j} + \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right)$$

Equation 2.10

$$H = E + \frac{p}{\rho}$$

Equation 2.11

$$\rho \frac{DH}{Dt} = \frac{Dp}{Dt} + \tau_{ij} \frac{\partial u_i}{\partial x_j} + \frac{\partial}{\partial x_{ji}} \left(k \frac{\partial T}{\partial x_j} \right)$$

2.2 Approximate Forms of the Equations of Motion

In a typical turbulent flow field, the variations in the velocity are small in relation to the speed of sound and the variations in the density are small in relation to the average density. As a result, the variations in the density are typically considered

negligible, and the mass density in the conservation equations is replaced by some average value. As a result, average values for the pressure and temperature of the fluid can be defined. Using these average values and defining the total heat as shown in Equation 2.12, the momentum equation can be written in the form

Equation 2.12

$$H = c_p T + \text{constant}$$

shown in Equation 2.13, and the energy equation can be written in the form shown in Equation 2.14. If p_a and ϕ are recognized as being independent of time, the

Equation 2.13

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = - \frac{\partial}{\partial x_i} \left(\frac{p - p_a}{\rho_z} \right) + \frac{T - T_a}{T_z} \frac{\partial \phi}{\partial x_i} + \frac{\partial}{\partial x_i} \left(\frac{\tau_{ij}}{\rho_a} \right)$$

Equation 2.14

$$\frac{DT}{Dt} = \frac{1}{\rho_a c_p} \left[\frac{D(p - p_a)}{Dt} + \frac{Dp_a}{Dt} \right] + \frac{\tau_{ij}}{\rho_a c_p} \frac{\partial u_i}{\partial x_j} + \frac{k}{\rho c_p} \frac{\partial^2 T}{\partial x_c^2}$$

fluctuations in pressure are assumed to be small, and the diffusivity is assumed to be small with respect to the specific heat, the energy equation can be written in the form of the potential energy equation, Equation 2.15.

Equation 2.15

$$\frac{D}{Dt} (T + \phi/c_p) = k \frac{\partial^2}{\partial x_i^2} (T + \phi/c_p)$$

Using the assumption of negligible oscillations in mass density and these forms of the energy and momentum equation, the set of equations describing the fluid motion can be reduced to the continuity equation and the equation of motion, shown in Equations 2.16 and 2.17, respectively (Townsend, 1976). Often, the buoyancy term is assumed negligible for forced flow and the equation of motion is

Equation 2.16

$$\frac{\partial u_i}{\partial x_i} = 0$$

Equation 2.17

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = - \frac{\partial}{\partial x_i} \frac{p - p_a}{\rho_a} + \nu \frac{\partial^2 u_i}{\partial x_j^2} + \frac{T - T_a}{T_a} \frac{\partial \phi}{\partial x_i}$$

expressed as shown in Equation 2.18. Equations 2.16, 2.18, and 2.15 are often referred to as the Boussinesq approximate form of the equations of motion.

Equation 2.18

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial}{\partial x_i} \left(\frac{p - p_a}{\rho_a} \right) + \nu \frac{\partial^2 u_i}{\partial x_j^2}$$

2.3 The Reynolds Equations of Turbulent Motion

The modeling of the turbulent fluctuations in a turbulent flow field begins with the development of the Reynolds equations, where any variable S is resolved into a mean value plus a fluctuation as shown by Equation 2.19. The mean value is

Equation 2.19

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

defined as shown in Equation 2.20, where T is assumed large in comparison to

Equation 2.20

$$\bar{S} = \frac{1}{T} \int_{t_0}^{t_0+T} S \, dt$$

the period of the fluctuations. Note that the mean value is allowed to fluctuate slowly over time itself, depending on the choice of period T . This case is considered an unsteady turbulent flow. For the incompressible turbulent flow with constant thermodynamic properties, possible fluctuations in velocity, pressure and temperature can be expressed using the Reynolds form as shown in Equation 2.21a-e.

Equation 2.21

$$\begin{array}{ll} u = \bar{u} + u' & p = \bar{p} + p' \\ \text{(a)} & \text{(d)} \\ v = \bar{v} + v' & T = \bar{T} + T' \\ \text{(b)} & \text{(e)} \\ w = \bar{w} + w' & \\ \text{(c)} & \end{array}$$

Assuming that the density of the fluid is constant and that the force of gravity is constant with any change in position, the conservation of mass, momentum and energy equations can be rewritten as the familiar continuity equation,

-Stokes equation, and energy equation – Equations 2.22 through 2.24. For

Equation 2.22

$$\vec{\nabla} \cdot \vec{V} = 0$$

Equation 2.23

$$\rho \frac{D\vec{V}}{Dt} = \rho \vec{g} - \vec{\nabla} p + \mu \nabla^2 \vec{V}$$

Equation 2.24

$$\rho c_p \frac{DT}{Dt} = k \nabla^2 T + \tau'_{ij} \frac{\partial u_i}{\partial x_j}$$

convenience, the vector form of the equation is shown rather than the implicit summation form from the previous section. Using the variables defined in Equation 2.21, the Reynolds form of the continuity equation indicates that the mean and fluctuating components of the velocity satisfy the continuity equation separately as shown in Equation 2.25a-b. In similar fashion, the Navier-Stokes equation can be

Equation 2.25

$$\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} = 0 \quad (a)$$

$$\frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} + \frac{\partial w'}{\partial z} = 0 \quad (b)$$

written in the Reynolds form shown in Equation 2.26, and the energy equation can be written in the Reynolds form shown in Equation 2.27.

Equation 2.26

$$\rho \frac{D\bar{V}}{Dt} + \rho \frac{\partial}{\partial x_j} (\overline{u'_i u'_j}) = \rho \vec{g} - \vec{\nabla} \bar{p} + \mu \nabla^2 \bar{V}$$

Equation 2.27

$$\rho c_p \frac{D\bar{T}}{Dt} = -\frac{\partial}{\partial x_i} \left(-k \frac{\partial \bar{T}}{\partial x_i} + \rho c_p \overline{u'_i T'} \right) + \frac{\mu}{2} \overline{\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial u'_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} + \frac{\partial u'_j}{\partial x_i} \right)^2}$$

The development of the basic equations of motion for the prediction of the typical, or time-averaged, flow behavior is relatively straightforward. The development of a model for the prediction of the behavior of the turbulent component from

first principles is not as easily achieved. The most obvious means of modeling the turbulent behavior is the addition of conservation of turbulence relations to the time-averaged continuity, momentum, and energy equations discussed. One means of developing such a relation is the consideration of the turbulent kinetic energy of the fluctuations, defined by Equation 2.28, where the implicit summation

Equation 2.28

$$K = \frac{1}{2} \left(\overline{u'u'} + \overline{v'v'} + \overline{w'w'} \right) = \frac{1}{2} u'_i u'_i$$

notation is used. The kinetic energy equation can be derived by formulating the dot product of u_i and the i th moment of the momentum equation, as seen previously in Equation 2.8, to get the mechanical energy equation. By subtracting the instantaneous mechanical energy equation from its time-averaged value, the turbulent kinetic energy equation for an incompressible fluid can be written as shown in Equation 2.29.

Equation 2.29

$$\begin{aligned} \frac{DK}{Dt} = & - \frac{\partial}{\partial x_i} \left[u'_i \left(\frac{1}{2} u'_j u'_j + \frac{p'}{\rho} \right) \right] - \overline{u'_i u'_j} \frac{\partial \bar{u}_j}{\partial x_i} \\ & + \frac{\partial}{\partial x_i} \left[v u'_j \left(\frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right) \right] - v \frac{\partial u'_j}{\partial x_i} \left(\frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right) \end{aligned}$$

While Equation 2.29 provides a relation for conservation of turbulence in the flow field, the terms in the equation are so complex that they cannot be computed from first principles. Furthermore, the turbulent kinetic energy proportional to the three turbulent normal stresses in the stress tensor, i.e. those components with repeated subscripts. Typically it is the shear stresses, or the components with unequal subscripts, that are of more importance to the development of mechanical systems. A relation can be developed to describe the conservation of any of the stresses, which are often called Reynolds stresses when the shear and normal stresses are considered to belong to a single group. The derivation of the equation requires that the time-averaged momentum equation be subtracted from its instantaneous form for both the i and j directions. The i th relation is then multiplied by u_j and the j th result by u_i . The sum of these relations is then time-averaged to derive the form of the Reynolds stress equation shown in Equation 2.30. Unfortunately, the terms in this relation are also so complex that they cannot be easily computed from first principles. Therefore, a more empirical modeling approach must be developed (White, 1991).

Equation 2.30

$$\begin{aligned} \frac{D \overline{u'_i u'_j}}{D t} = & - \left[\overline{u'_j u'_k} \frac{\partial \bar{u}_i}{\partial x_k} + \overline{u'_i u'_k} \frac{\partial \bar{u}_j}{\partial x_k} \right] - 2 \overline{v \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k}} \\ & + \frac{p'}{\rho} \left(\frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right) - \frac{\partial}{\partial x_k} \left[\overline{u'_i u'_j u'_k} - v \frac{\partial \overline{u'_i u'_j}}{\partial x_k} + \frac{p'}{\rho} (\delta_{jk} u'_i + \delta_{ik} u'_j) \right] \end{aligned}$$

2.4 Turbulence Modeling

While the Reynolds stress equation and the turbulent kinetic energy equation provide a means of evaluating the turbulent behavior of a flow field, they are not practical to use because of the complexity of the terms in both equations. Thus turbulence is necessary to complete the evaluation of the flow field. If the buoyancy and the temperature dependence of the fluid properties are neglected, the energy equation may be uncoupled to allow the solution of the temperature distribution to be performed after the velocity distribution is well known. Thus the continuity and momentum equations can be considered the primary equations of motion. The approach to adding turbulence modeling to these equations may be divided into six categories:

1. An eddy viscosity approximation is directly applied to the momentum equation.
2. One equation, usually some form of the turbulent kinetic energy equation, is added to the set.
3. Two equations, usually the turbulent kinetic energy equation and a model of the dissipation, turbulence length scale, or vorticity fluctuations, are added to the set.
4. A set of Reynolds stress models is added to the original set of two equations.
5. Large-eddy simulation, which is an almost model free approach, is used in conjunction with the two primary equations.
6. Direct numerical simulation is used to evaluate turbulence in a completely model free approach.

Models that fit in the first category are very limited in their range of application and are only reliable in simple flow fields. Models that fit in the fifth and sixth categories are computationally intensive and simply not practical for large scale applications (White, 1991). Therefore, the remainder of this chapter will focus on

the more commonly used schemes that are found in the second, third and fourth categories.

One-Equation Turbulence Models

For convenience the turbulence kinetic energy equation is usually considered in the boundary-layer form shown in Equation 2.31. If the effective eddy size is

Equation 2.31

$$\bar{u} \frac{\partial K}{\partial x} + \bar{v} \frac{\partial K}{\partial y} \approx - \frac{\partial}{\partial y} \left[\bar{v}' \left(\frac{1}{2} \bar{u}'_i \bar{u}'_i + \frac{p'}{\rho} \right) \right] + \frac{\tau}{\rho} \frac{\partial \bar{u}}{\partial y} - \varepsilon$$

defined as L , and the velocity scale of such an eddy is assumed to be $K^{1/2}$ then the dissipation, ε , should scale by the rule shown in Equation 2.32. This makes sense

Equation 2.32

$$\varepsilon \approx (\text{constant}) \frac{K^{3/2}}{L}$$

physically if one considers that the power dissipated per unit mass by an eddy of size L moving through a fluid with speed $K^{1/2}$ must be proportional to the relation given in Equation 2.33. Thus, the third term on the right hand side of Equation

Equation 2.33

$$\frac{\text{drag} \times \text{velocity}}{\text{mass}} \approx \frac{[(\text{constant}) K L^2] K^{1/2}}{(\text{constant}) L^3} \approx (\text{constant}) \frac{K^{3/2}}{L}$$

2.31 can be evaluated. The second term on the right hand side of Equation 2.31 is the turbulent kinetic energy production term and is modeled by a replication of the eddy viscosity model from the first category of approaches to turbulence modeling. The typical formulation of the eddy viscosity model is shown in Equation 2.34.

Equation 2.34

$$\tau = (\mu + \mu_t) \left(\frac{\partial \bar{u}}{\partial y} \right)$$

The first term on the right hand side, the convective diffusion term, is approached by assuming this term is similar to gradient diffusion as indicated by Equation 2.35. Thus, a generic model for the turbulent kinetic energy can be developed as shown in Equation 2.36. This model is not commonly used, however, because

it offers only slight improvements over the use of the eddy viscosity directly in the momentum equation (White, 1991).

Equation 2.35

$$-\overline{v' \left(\frac{1}{2} u_i' u_i' + \frac{p'}{\rho} \right)} \approx (\text{constant}) \frac{\partial K}{\partial y}$$

Equation 2.36

$$\bar{u} \frac{\partial K}{\partial x} + \bar{v} \frac{\partial K}{\partial y} \approx \frac{\partial}{\partial y} \left[(\text{constant}) \frac{\partial K}{\partial y} \right] + v_t \left(\frac{\partial \bar{u}}{\partial y} \right)^2 - (\text{constant}) \frac{K^{3/2}}{L}$$

Two-Equation Turbulence Models

The turbulent kinetic energy equation provides better results when coupled with a second equation that models the rate of change of the dissipation or turbulent length scale. The dissipation approach is the most commonly used, probably because the approach is more intuitive. Models of this type are called K - ϵ models. The forms of the two equations that are most commonly used are those that apply to fully elliptic, or non-boundary layer, high-Reynolds-number flow, as shown in Equations 2.37 and 2.38. The eddy viscosity itself appears in both equations and

Equation 2.37

$$\frac{DK}{Dt} \approx \frac{\partial}{\partial x_j} \left(\frac{v_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + v_t \frac{\partial \bar{u}_i}{\partial x_j} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \epsilon$$

Equation 2.38

$$\frac{D\epsilon}{Dt} \approx \frac{\partial}{\partial x_j} \left(\frac{v_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right) + C_1 v_t \frac{\epsilon}{K} \frac{\partial \bar{u}_i}{\partial x_j} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - C_2 \frac{\epsilon^2}{K}$$

is modeled as shown in Equation 2.39. The unknown constants in these equations

Equation 2.39

$$v_t \approx \frac{C_3 K^2}{\epsilon}$$

must determined empirically for specific applications. Along with the equations of continuity and momentum, these two equations form the K - ϵ model. These models neglect molecular viscosity and sub-layer dampening effects. Therefore, they cannot be used in the near wall region, where a log law relation is often

assumed. The $K - \epsilon$ calculation begins at some point in the near wall region where the boundary conditions are provided by the log-law assumption (White, 1991).

Reynolds Stress Models

The modeling of Reynolds stresses goes a step beyond the models discussed previously. In this case, the approximations inherent in the eddy viscosity and velocity gradient approach are abandoned and the stresses are computed directly. As a result, this type of modeling is computationally intensive and often not practical for large systems. The basic approach is to consider individually each of the terms in the Reynolds stress equation, Equation 2.30. Each term is modeled using empirical relations appropriate for the individual application (White, 1991). A large and diverse group of empirical relations have been developed, and new ones continuously appear in the literature. While the Reynolds stress model approach theoretically allows more flexibility, it is currently difficult to apply to complex systems. For this reason, the simpler $K - \epsilon$ model is more commonly used in engineering development.

Chapter 3 Description of Experimental Facilities

The preliminary Spallation Neutron Source (SNS) flow experiments focus on the characterization of the dynamic behavior of the flow field for verification of the computational model used to predict temperature distributions within the mercury target. These experiments use air and water as surrogate fluids for the liquid mercury. The use of a surrogate fluid eliminates the need for safety systems to protect operators from liquid mercury and mercury vapor exposure, reduces corrosion and material compatibility concerns, and facilitates the use of flow visualization techniques and Laser Doppler Velocimetry (LDV) to characterize the flow field. However, the use of a surrogate fluid requires that the experimental plans be properly scaled to ensure similarity between the scaled experiment and the actual system. Two scaled experimental facilities are available for flow characterization studies in support of the SNS target design: the University of Tennessee Air Test Facility (UTATF) and the Water Thermal Hydraulic Loop (WTHL) at Oak Ridge National Laboratory.

3.1 University of Tennessee Air Test Facility

The UTATF, shown in Figure 3.1, is a scaled flow experiment facility that uses air as a surrogate fluid for liquid mercury. The UTATF test section, shown in Figure 3.2, is a 1.2-scale two-dimensional representation of the SNS target bulk flow that uses dimensions from the target's horizontal midplane. The upper surface of the UTATF test section is manufactured of clear acrylic to allow direct observation of flow behavior. Air is drawn through the system by suction to a blower at the exit, with airflow velocities modified by throttling the blower and attaching orifice plates at the inlets. The nominal flow rate for UTATF experiments is 17 l/s per inlet leg. A single pressure transducer and a hot wire anemometer are available for local pressure and velocity measurements,

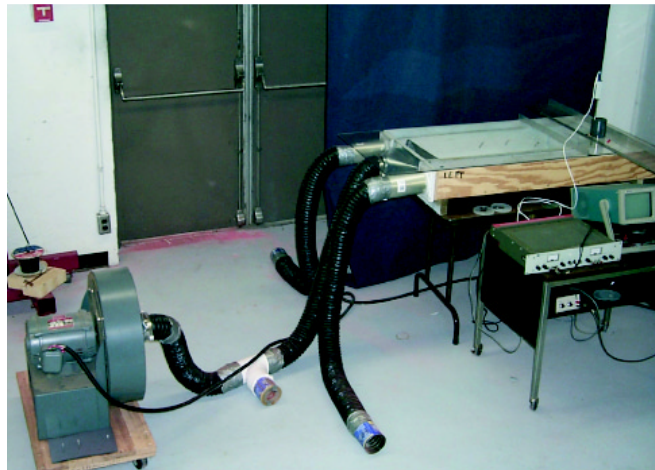


Figure 3.1 The University of Tennessee Air Test Facility.

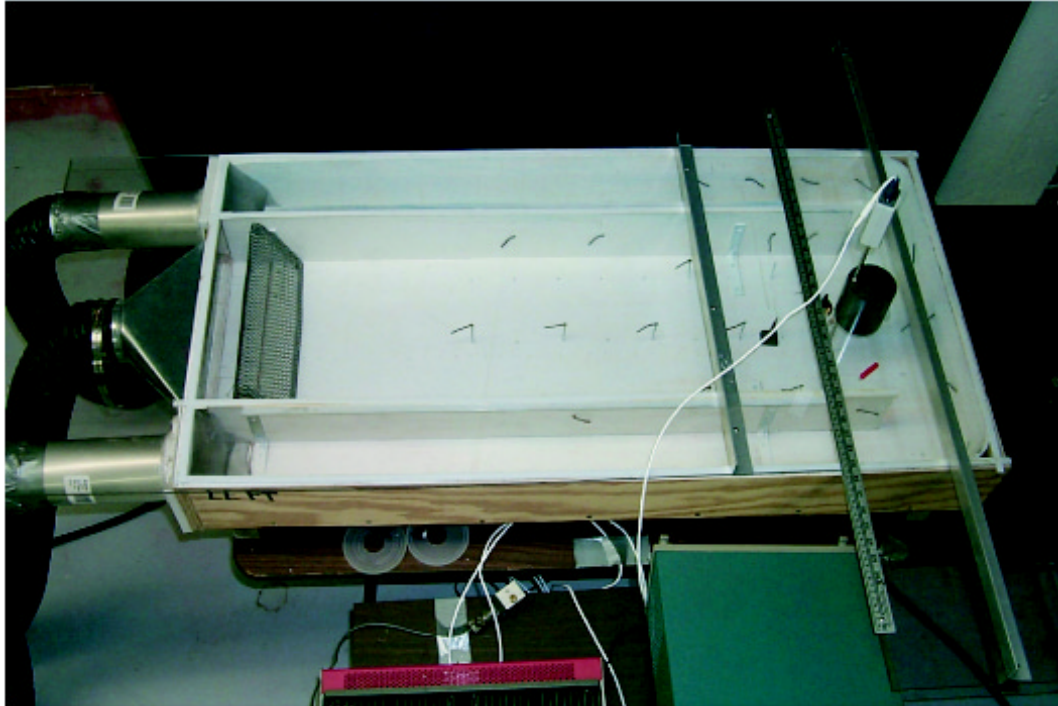


Figure 3.2 – Test section of the University of Tennessee Air Test Facility.

and smoke injection is used for flow visualization. The primary purpose of the UTATF experiments is the identification of significant flow phenomena for further evaluation in the more rigorous WTHL experiments.

3.2 The Water Thermal Hydraulic Loop

The Water Thermal Hydraulic Loop (WTHL) is a scaled experiment facility for evaluation of flow patterns in the bulk mercury flow regions of the SNS mercury target. The WTHL uses water as a surrogate fluid for liquid mercury. The use of water as a surrogate fluid for liquid mercury has several benefits in a facility designed for characterization of a fluid flow field. Since liquid mercury is completely opaque, optical methods of flow field evaluation cannot be used in a liquid mercury facility. The use of water as a surrogate fluid for liquid mercury allows the use of flow visualization techniques for evaluation of bulk flow patterns and optical techniques for localized flow velocity measurements. In liquid mercury facilities, safety systems and procedures must be in place to protect personnel from exposure to liquid mercury or mercury vapor. The use of water as a surrogate fluid eliminates the need for such precautions. Furthermore, liquid mercury is highly corrosive to many materials, requiring careful selection of components to ensure the integrity of the system. The use of water as a surrogate fluid also

alleviates these restrictions.

A schematic of the WTHL is shown in Figure 3.3. The facility contains roughly 500 gallons of water distributed between a 300-gallon tank, flow lines and the test section. The majority of the flow lines are constructed from 8-inch PolyVinyl Chloride (PVC) piping, although some carbon and stainless steel components are used in the two inlet legs that feed water into the test section. The system can be configured to flow through the tank or bypass the tank as needed. The facility is powered by a 15-horsepower variable speed pump that can provide flow rates through the system as a high as 40 L/s (or 20 L/s per inlet leg). The nominal flow rate for the WTHL experiments is 22 L/s (or 11 L/s per inlet leg).

The flow rate through the test section is monitored by a magnetic flow meter on each inlet leg. Three pressure gauges and five pressure transducers are used to monitor local pressures in the inlet legs and the test section. A display screen, developed using the Intellution software package, is used to monitor

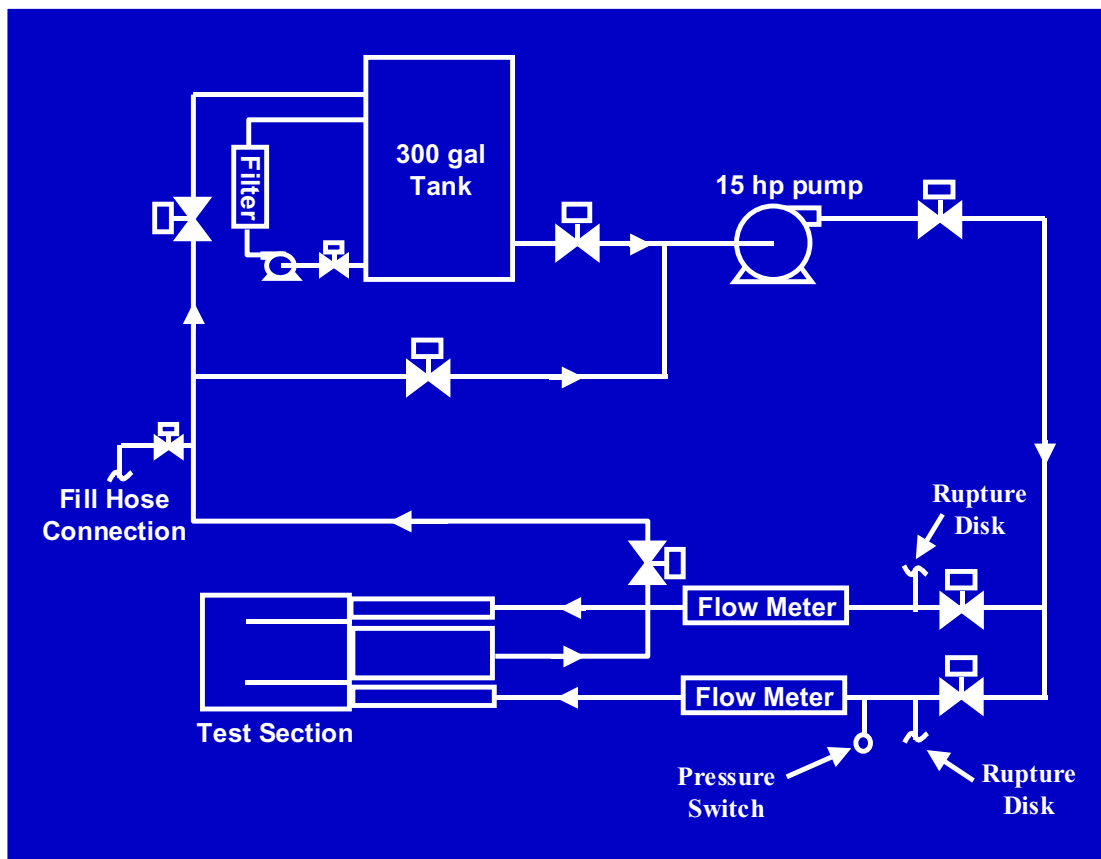


Figure 3.3 – Schematic of the WTHL.

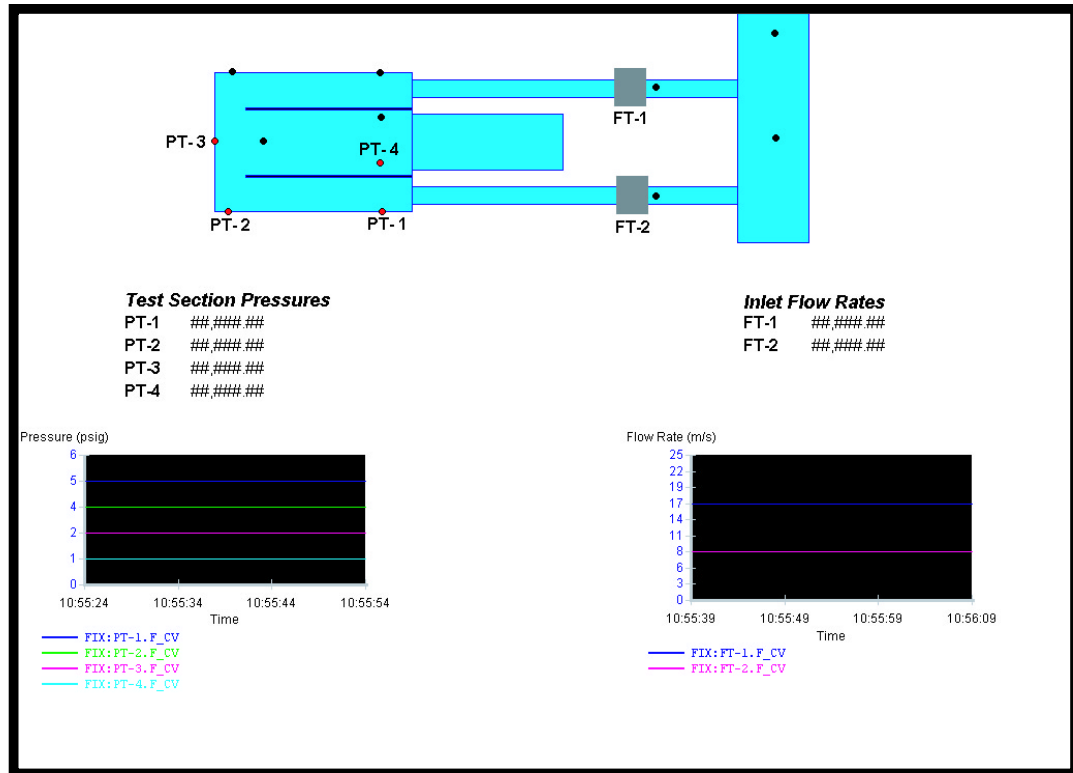


Figure 3.4 Sample screen from WTHL data collection system.

and record flow rate and local pressure measurements registered by the facility instrumentation. A sample display screen is shown in Figure 3.4.

The WTHL uses a test section that is an exact scale model of the bulk flow regions of the nominal mercury target design. The test section does not include the mercury window cooling channels that provide additional heat transfer for cooling of the target nose where the proton beam enters the target. All test sections to be used in the WTHL experiments are manufactured of a clear material to allow flow visualization studies using bubble or dye injection at various points in the flow field. The nominal WTHL test section, shown in Figure 3.5 and Figure 3.6, is manufactured in two pieces that are bolted together and then bolted to the stainless

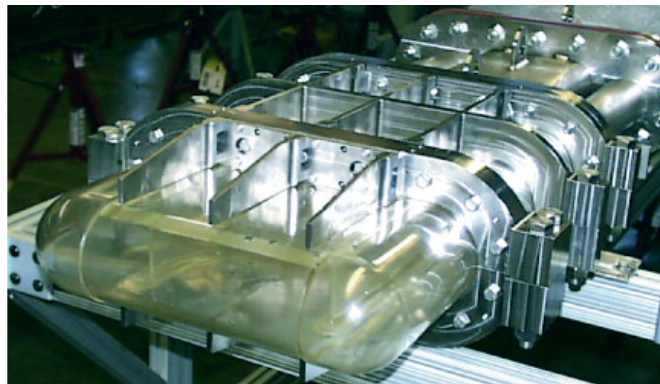


Figure 3.5 – Water Thermal Hydraulic Loop Test Section.

steel inlet and exit spool piece that feed water into the test section as shown in Figure 3.7. The clear portions of the test section were fabricated using stereo lithography to produce molds from a three-dimensional solid model of the SNS target. These portions of test section are then molded using Hapco's Liquid Molding Compound 406.

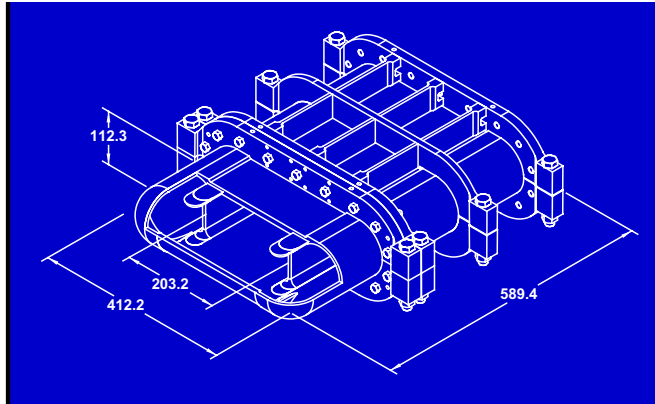


Figure 3.6 – Schematic of the WTHL test section. Dimensions shown in millimeters.

Two systems are included in the WTHL to protect the integrity of the test section. A pressure switch that removes power to the pump when the pressure is greater

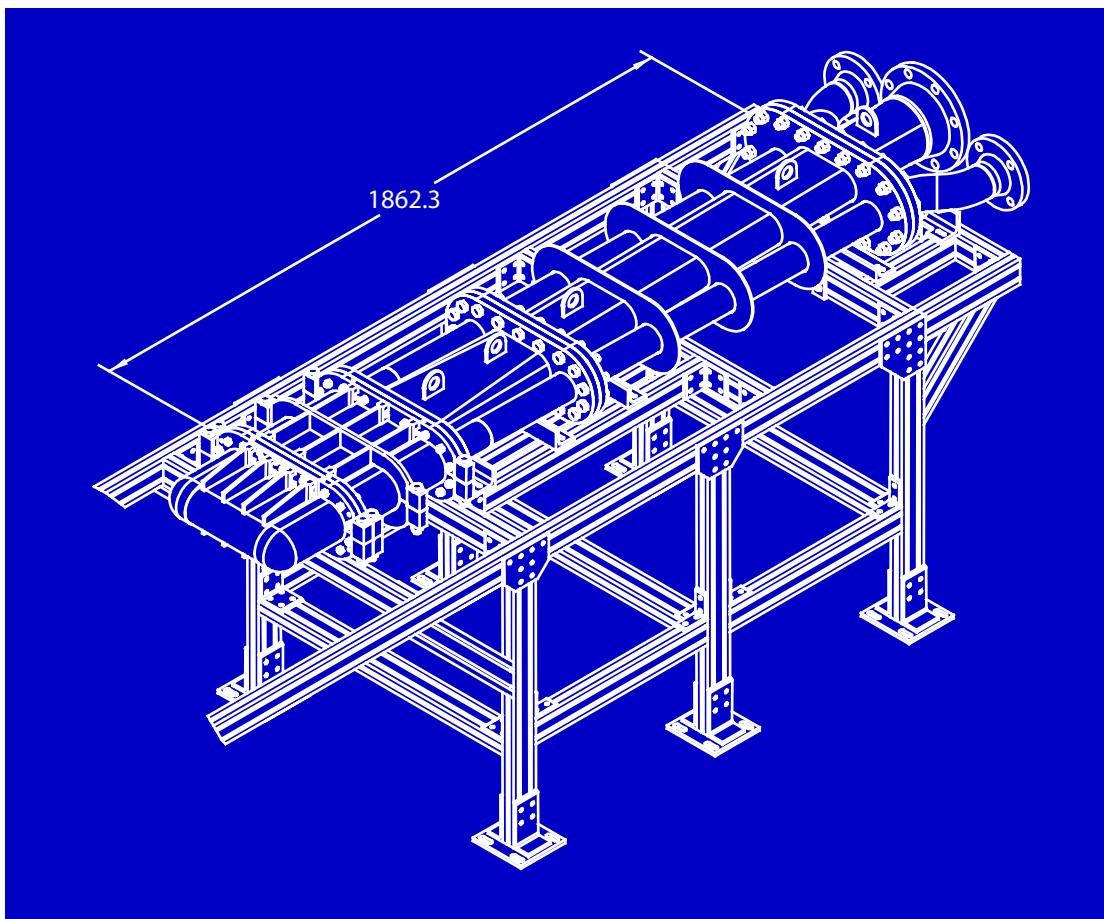


Figure 3.7 Schematic of the WTHL test section and spool piece. Dimensions shown in millimeters.

than the set point pressure is included in one of the two inlet legs. The set point pressure is set to 12 psig for all WTHL experiments to prevent deformation of the WTHL test section. Additionally, a 25 psig rupture disk and pressure relief line are included in each inlet leg to prevent damage to the test section in the event of pressure switch failure.

The primary instrument for the mapping of the velocity field within the mercury target bulk flow region is a two-beam Laser Doppler Velocimeter (LDV), manufactured by TSI, Inc., shown in Figure 3.8. The system uses the green and blue lines of an Argon laser to measure two components of the flow velocity within a measurement volume that is roughly 1 mm^3 . Two-beam LDV systems focus two pairs of laser beams (i.e., two blue and two green laser beams) on a common point in the flow field. An interference or fringe pattern that defines the measurement volume forms at the intersection of the beams. Particles moving through the measurement volume scatter light that is collected by a photodetector. The frequency of the photodetector output is directly related to the velocity of the particle that passed through the measurement volume. When two pairs of laser beams are focused on the measurement volume, two velocity components can be measured

simultaneously. Furthermore, if the frequency of one of the beams in each pair is shifted, the technique can detect flow direction (plus or minus relative to direction of the vector connecting the shifted and unshifted beams) in addition to flow velocity. LDV measurements are single-point measurements. Area analyses are performed using a traverse system to move the light source from point to point (Laser Doppler Velocimetry Catalog, 2000). The major advantage of this technique of velocity measurement is that the measurement is non-intrusive and does not introduce artificial flow patterns into the flow field as the measurement is performed.

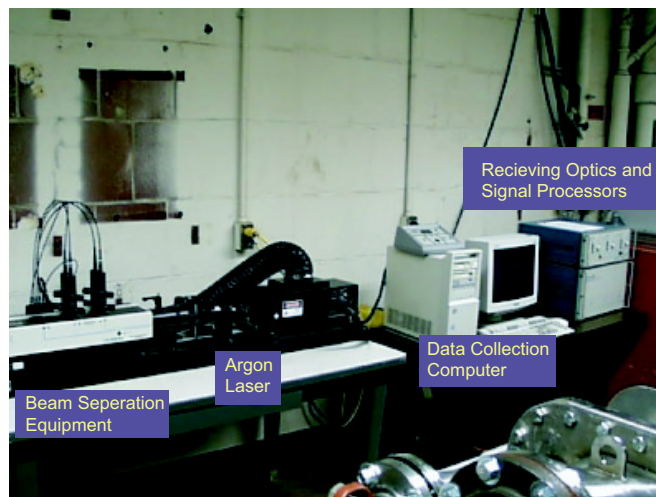


Figure 3.8 – Two-beam Laser Doppler Velocimeter equipment. Shown are Argon laser and associated multi-color beam separation equipment, receiving optics and signal processors, and personal computer for data collection. The LDV probe, not visible in this picture, is mounted over the WTHL test section.

In order to obtain reliable LDV measurements, the surface of the container material must be free of any imperfections that may refract or reflect any of the four laser beams such that the beams do not cross through a single volume at the measurement location. For this reason, test sections to be used for LDV measurements are modified to include a 120 mm by 160 mm Plexiglas window near the nose region of the test section and/or a 215 mm by 160 mm Plexiglas window in the exit channel of the rear test section piece. A test section with both windows installed is shown in Figure 3.9. Figure 3.10 shows a close-up view of the front window.

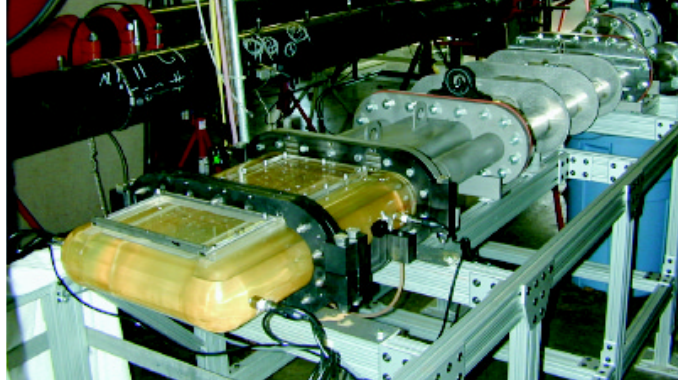


Figure 3.9 – WTHL test section with front and rear Plexiglas windows for LDV measurements.

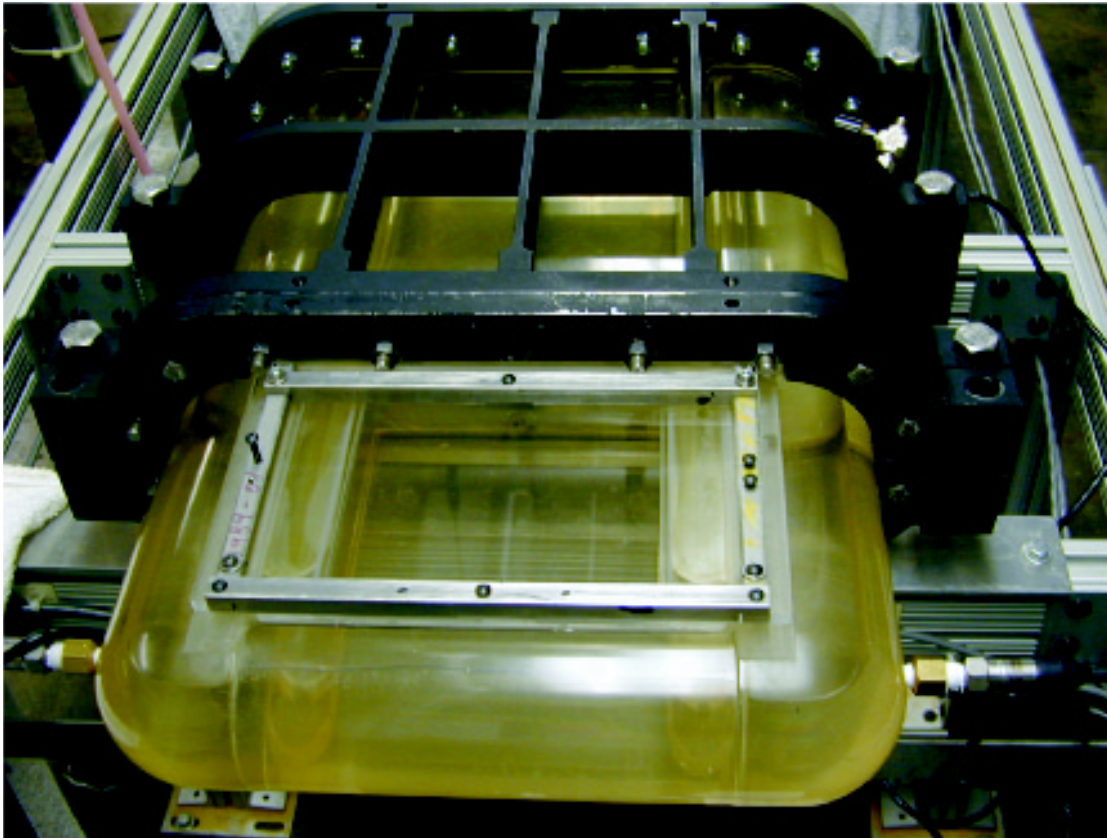


Figure 3.10 – Close-up view of WTHL test section with single Plexiglas window.

An Ultrasonic Doppler Profiliometer (UVP) is also available in the WTHL experiments, providing a second instrument for the mapping of velocities in the test section. The UVP probe emits a sound wave of ultrasonic frequency (1 MHz in the case of the WTHL experiments) into a flow field that has been seeded with small bubbles that are fractions of a millimeter in diameter. The sound wave is reflected back to the probe by the scattering centers created by the bubbles. The probe records the reflected signal and evaluates the velocity of the scattering by measuring the Doppler shift in the frequency of the wave while simultaneously recording the position of the scattering center by measuring the time between emission of the initial wave and detection of the reflected wave. The primary advantages of this technique are the ability to map velocity fields in opaque fluids or containers and the ability to obtain simultaneous measurements of velocity and position. The primary disadvantage is that the uncertainty in position increases as the distance from the probe increases because the diameter of the ultrasonic beam expands as it moves through the fluid. The primary purpose for the use of this instrument in these experiments is the qualification of its abilities for use in future prototypic mercury experiments.

Chapter 4 Similarity and Scaling Studies

Supporting the Experimental Program

With any scaled experimental program, it is important to evaluate the validity of the data collected for application to the real-world system. Uncertainty analyses bound the precision with which measurements are taken and help to establish the reliability of a data set. However, they do not provide any information regarding the usefulness of the acquired data to a specific application. Similarity studies provide a measure of the usefulness of a data set to a specific application as well as a basis for extrapolation of the experimental results to that application.

In similarity studies for fluid systems, scaling parameters that relate to important flow phenomena are evaluated for both the experimental facility and the application. The most common scaling parameters are typically dimensionless numbers that appear as coefficients in non-dimensionalized forms of any of the conservation equations that govern the motion of fluid through the system. Thus, scaling parameters are usually functions of thermophysical properties of the fluid and the physical specifications of the flow field. Furthermore, these parameters are typically representative of the importance of a specific type of force acting on the fluid.

4.1 Similarity of Planar Forces and Stresses

Planar forces and stresses occur in all mechanical systems. They are an engineering approach to representing the resistance of an object to changes in its shape. In bulk systems, they are represented by coefficients, such as the viscosity in fluid mechanics. On the differential scale, they are represented by stress tensors acting on the differential element. In fluid mechanics, these forces are responsible for the development of streamline patterns in laminar flows and for the development of turbulence structures in turbulent flows. Therefore, it is imperative that a scaled experiment designed to examine the characteristic flow patterns in a flow field have planar forces and stress that are similar to those in the system being simulated.

Derivation of the Reynolds Number

If the variables that appear in the conservation equations are redefined as dimensionless quantities, the conservation equations can be written in a non-dimensional form. Then, each term in the equation takes the form of a dimensionless

variable component of order unity that is multiplied by a dimensionless constant. The magnitude of the constant multiplying the variable component can be used to evaluate the significance of that term in relation to the other terms in the equation. The dependent and independent variables are typically non-dimensionalized by dividing the variable by some reference property as shown in Equations 4.1a-f.

Equation 4.1

$$x_i^* = \frac{x_i}{L} \quad p^* = \frac{p}{\rho U_\infty^2}$$

(a) (b)

$$u^* = \frac{u}{U_\infty} \quad T^* = \frac{T - T_\infty}{T_w - T_\infty}$$

(c) (d)

$$t^* = \frac{t U_\infty}{L} \quad \vec{\nabla}^* = L \vec{\nabla}$$

(e) (f)

The continuity equation for an incompressible fluid is given by Equation 4.2. Using

Equation 4.2

$$\vec{\nabla} \cdot \vec{v} = 0$$

the dimensionless variables defined in Equation 4.1, the continuity equation can be written in the dimensionless form shown in Equation 4.3. While this result appears rather uninteresting since the equation does not change, it does indicate that the continuity relation is independent of the flow parameters and fluid properties.

Equation 4.3

$$\vec{\nabla}^* \cdot \vec{v}^* = 0$$

The conservation of momentum (Navier-Stokes) equation is given by Equation 4.4. For most flows, the effects of gravity are negligible (the effects of gravity in the SNS target are discussed in detail in Section 4.2) and the equation can

Equation 4.4

$$\rho \frac{D\vec{v}}{Dt} = \rho \vec{g} - \vec{\nabla} p + \mu \nabla^2 \vec{v}$$

be written as Equation 4.5. The dimensionless form of the momentum equation

Equation 4.5

$$\rho \frac{D\vec{v}}{Dt} = -\vec{\nabla} p + \vec{\nabla} \cdot \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]$$

can be obtained by substituting the dimensionless forms of the dependent and independent variables (Equation 4.1) into Equation 4.5. The dimensionless form is shown in Equation 4.6 where the Reynolds number, Re , is defined in Equation 4.7.

Equation 4.6

$$\frac{D\vec{v}^*}{Dt^*} = -\vec{\nabla}^* p^* + \frac{1}{Re} \vec{\nabla}^* \cdot \left[\left(\frac{\partial u_i^*}{\partial x_j^*} + \frac{\partial u_j^*}{\partial x_i^*} \right) \right]$$

Equation 4.7

$$Re = \frac{\rho U_\infty L}{\mu}$$

Since the other terms in the momentum equation are on the order of unity, the Reynolds number can be viewed as a ratio of the forces resulting from the inertia of the fluid to the forces resulting from the viscous stresses acting on the fluid. Thus, the Reynolds number is a dimensionless scaling parameter that indicates the relative importance of the viscous stresses to the general behavior of the flow. When the Reynolds number is large, i.e. greater than approximately 10^4 , the flow field will exhibit turbulent flow characteristics. When it is small, the flow field will exhibit the smooth streamlines characteristic of laminar flow.

Evaluation of the Reynolds Number for the SNS Thermal Hydraulic Experiments

The University of Tennessee Air Test Facility (UTATF) experiments are intended to provide qualitative insight into the behavior of the bulk flow region of the liquid mercury target. Therefore, a high degree of similarity between the two systems is not necessary. However, the results of the UTATF experiments are only valid if there is some similarity between the two systems. The Reynolds numbers in the nominal mercury target and the UTATF are 1.45×10^6 and 2.00×10^4 , respectively. While the Reynolds numbers in the two systems do not match, both are well within the turbulent range, indicating that some similarity can be expected between the

two systems.

The Water Thermal Hydraulic Loop (WTHL) experiments are intended to characterize the flow behavior of the liquid mercury target. Thus, the WTHL experiments must mimic the flow behavior in the bulk mercury flow of the reference target design as closely as reasonably achievable. The Reynolds Number is shown in Figure 4.1 for the WTHL test section and the nominal bulk flow rate of the Spallation Neutron Source (SNS) mercury target as a function of volumetric flow rate. Clearly, at least five times the available flow rate (20 l/s) is required to match the Reynolds number of the nominal mercury target. The Reynolds Number for the nominal WTHL experiments is 1.39×10^5 . Thus, the Reynolds number for the WTHL experiments is still well within the turbulent range, indicating that adequate similarity can be achieved in those experiments.

Turbulent Fluid Flow

Scaling the viscous forces in the SNS thermal hydraulic experiments is important because these forces strongly influence the development of turbulence in the flow field. For flow in a pipe, turbulence begins to develop for Reynolds numbers greater than about 2300 and is fully developed for Reynolds numbers greater than 10,000. The development of turbulence dramatically changes the nature of the

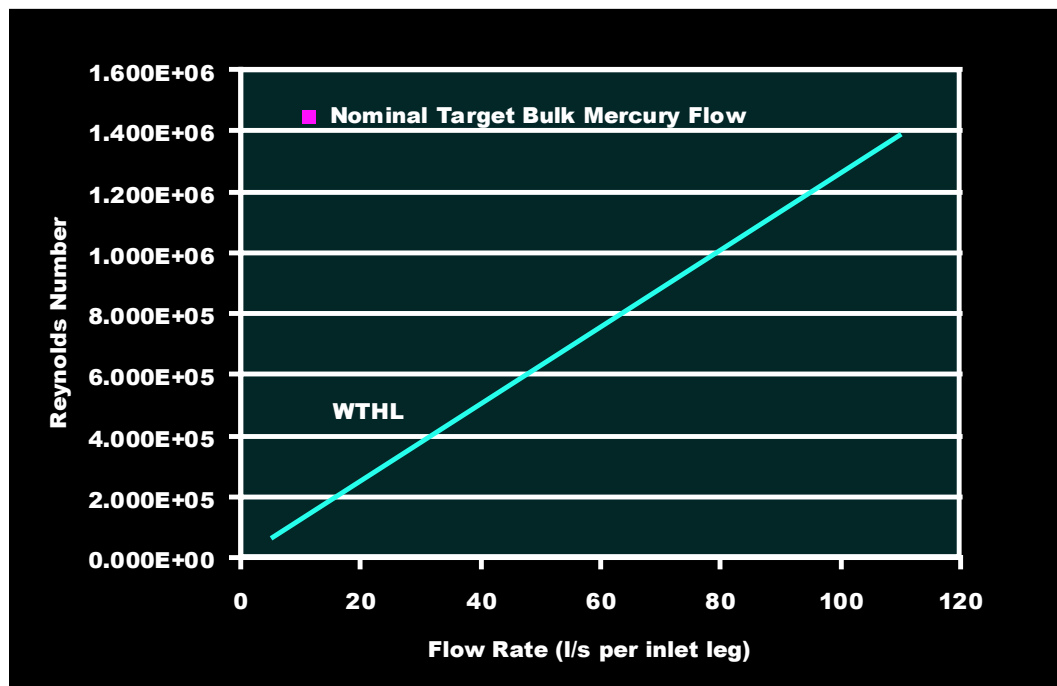


Figure 4.1 – Reynolds Number in the SNS Mercury Target and the WTHL.

flow. The mixing that occurs in turbulent flow typically results in an increase in heat transfer, transmission of shear stresses, and mass diffusion. For example, the shear stress in parallel laminar flow ($Re < 2300$) is given by Newton's law of friction, shown in Equation 4.8. For parallel turbulent flow, Newton's law of friction is

Equation 4.8

$$\tau = \mu \frac{\partial u}{\partial y} = \rho \nu \frac{\partial u}{\partial y}$$

commonly rewritten using Boussinesq's eddy viscosity formulation. Boussinesq's equation is shown in Equation 4.9 where ϵ_m , the eddy viscosity or the turbulent

Equation 4.9

$$\tau_t = \rho (\nu + \epsilon_m) \frac{\partial u}{\partial y}$$

diffusivity of momentum, is the effective increase in the shear stress resulting from turbulence in the flow field. While the Reynolds number can be used to determine if a flow is turbulent, it does not necessarily indicate the level of turbulence present. Two other flow parameters are better suited for the scaling of the level of turbulence in the flow field – turbulence intensity, Ti , and turbulent kinetic energy (Reynolds, 1974).

Turbulence Intensity

Considering some point in a turbulent flow that is steady, the three components of the instantaneous velocity vector can be written as shown in Equation 4.10,

Equation 4.10

$$\begin{aligned} u_1 &= U + u' \\ u_2 &= U + v' \\ u_3 &= U + w' \end{aligned}$$

where U is the time averaged velocity, u' is the velocity fluctuation in the mean flow direction, and v' and w' are velocity fluctuations normal to the mean flow direction. Clearly, the time averaged value of the fluctuations is zero as indicated in Equation 4.11. Thus, the intensity of each velocity fluctuation is given by the average of

Equation 4.11

$$\overline{u'} = \overline{v'} = \overline{w'} = 0$$

the square of the fluctuation, indicated as shown in Equation 4.12, and the overall

Equation 4.12

$$\overline{u'^2}, \quad \overline{v'^2}, \quad \overline{w'^2}$$

intensity of the fluctuation is given by Equation 4.13. The relative intensity of the

Equation 4.13

$$\overline{q^2} = \overline{u'^2} + \overline{v'^2} + \overline{w'^2}$$

turbulence may be measured by the dimensionless parameter commonly called the turbulence intensity, shown in Equation 4.14. Unlike the Reynolds number,

Equation 4.14

$$Ti = \frac{\left(\frac{1}{3}\overline{q^2}\right)^{1/2}}{U}$$

the turbulence intensity is directly related to the level of mixing that occurs within the flow field and, hence, the effective increase in the heat transfer, shear stress transmission, and mass diffusion (Reynolds, 1974).

While the turbulence intensity is commonly measured as part of any flow experiment exhibiting turbulent behavior, few practical models exist for the calculation of the turbulence intensity in a flow field. Most efforts to characterize the development of turbulence have focused on the turbulent boundary layer that develops as the fluid moves across a plate or body in cross-flow.

In order to develop models to evaluate turbulent boundary layer flow, the turbulent shear stress in the boundary layer must be modeled to close the system of equations formed by the conservation equations. Two formulations have historically provided this closing function:

1. Boussinesq's eddy viscosity formulation (Equation 4.9), which is analogous to the laminar flow.
2. Prandtl's mixing length formulation, which is of the form shown in Equation 4.15, where l_m is the Prandtl mixing length.

Equation 4.15

$$\tau_t = \rho l_m^2 \left| \frac{\partial U}{\partial y} \right| \frac{\partial U}{\partial y}$$

The mixing length model follows more closely with the physical interpretation of the shear stress behavior in turbulent boundary layers. The mixing length is somewhat analogous to the mean free path between molecular collisions. However, the mixing length is an effective interaction distance between elements of fluid rather than molecules.

In keeping with the analogy to the laminar flow form, the turbulent dynamic viscosity coefficient can be defined in terms of the mixing length as shown in Equation 4.16. If the mixing length, l_m , is assumed to be the distance that a fluid element

Equation 4.16

$$\mu_t = \rho l_m^2 \left| \frac{\partial U}{\partial y} \right|$$

travels, the element will induce a velocity perturbation given by Equation 4.17. For

Equation 4.17

$$u'(y_0) = U(y_0 + l_m) - U(y_0) \approx l_m \frac{\partial U}{\partial y}$$

fully developed turbulent internal flows, the Prandtl mixing length can be evaluated based on mean flow models. Mean flow models imply that the local shear stress can be represented as shown in Equation 4.18. Substituting the definition of l_m into

Equation 4.18

$$\tau = \tau_w \left(1 - \frac{y}{R} \right) - \tau_t$$

the shear stress equation and integrating using the measured logarithmic velocity profile for flow in a pipe, the mixing length can be found as a function of the pipe radius, R , and position, y , as shown in Equation 4.19. Thus, assuming fully developed turbulent flow in a pipe, the turbulence intensity, Ti , can be estimated using Equations 4.19 and 4.17, if the velocity gradient in the pipe is known (Schetz, 1993).

Equation 4.19

$$l_m = 0.14 R - 0.08 R \left(1 - \frac{y}{R} \right)^2 - 0.06 R \left(1 - \frac{y}{R} \right)^4$$

Based on extensive measurements in boundary layers developing along the surface of cylinders in cross-flow, Zhukauskas and Zhiugzhda (1985) propose a model for the turbulent dynamic viscosity of the form shown in Equation 4.20,

Equation 4.20

$$\mu_t = K \text{Ti} \delta U_\infty n^*$$

where the K is a universal constant ($K=0.15$) and the multiplier n^* is given by Equation 4.21. A model of this form does not require as much knowledge of the flow field as a model of the mixing length form.

Equation 4.21

$$n^* = [1 - \cos(\pi y / \delta)] / 2$$

Several turbulence intensity models have been developed based on velocity fluctuation measurements in a turbulent boundary layer developing over a flat plate. For the turbulent (outer) region of a turbulent boundary layer developing as fluid flows across a flat plate, Perry, Henbest and Chong (1986) demonstrated that the turbulence intensity in the direction of flow can be estimated using a model of the form shown in Equation 4.22. In this form, A_1 and C are universal constants

Equation 4.22

$$\frac{\overline{u_1^2}}{U_\tau^2} = B_1 - A_1 \ln \left| \frac{y}{\delta} \right| - C y_+^{-1/2}$$

and B_1 is a large-scale effect parameter. The wall shear velocity, U_τ , is given by Equation 4.23 and the length scale, y_+ , is given by Equation 4.24. Perry and Li

Equation 4.23

$$U_\tau = \left(\frac{\tau_w}{\rho} \right)^{1/2}$$

Equation 4.24

$$y_+ = \frac{y U_\tau}{\nu}$$

(1990) suggested an improvement to the viscous correction of the form shown in Equation 4.25, with the function V defined by Equation 4.26. Spalart (1988) had earlier proposed a simpler relation for the viscous correction of the form shown in Equation 4.27. Perry and Li show good agreement with experimental data for y_+

Equation 4.25

$$\frac{\overline{u_1^2}}{U_\tau^2} = B_1 - A_1 \ln \left| \frac{y}{\delta} \right| - V(y_+)$$

Equation 4.26

$$V(y_+) = 5.58(y_+)^{-\frac{1}{2}} - 22.4(y_+)^{-1} + 22.0(y_+)^{-\frac{5}{4}} - 5.62(y_+)^{-2} + 1.27(y_+)^{-\frac{1}{4}}$$

Equation 4.27

$$V(y_+) = 4.37(y_+)^{-\frac{1}{2}}$$

greater than 200 using values for the constants of $A_1=1.03$ and $B_1=2.39$.

Perry and Li proposed a numerical approach that would produce a more exact solution for the viscous correction term. The resulting formulation (as published by Marusik, Uddin, and Perry (1997)) is given by Equation 4.28. Marusik, Uddin

Equation 4.28

$$V(y_+) = 5.58 \left[1 - y_+^{-0.9} \right] y_+^{-0.5}$$

and Perry proposed an extended formulation of the form shown in Equation 4.29,

Equation 4.29

$$\frac{\overline{u_1^2}}{U_\tau^2} = B_1 - A_1 \ln \left| \frac{y}{\delta} \right| - V_g \left(y_+, \frac{y}{\delta} \right) - W_g \left(y_+, \frac{y}{\delta} \right)$$

where V_g is a viscous correction, and W_g is a wake correction. The viscous correction is given by Equation 4.30. The switching functions S_A and S_V ensure

Equation 4.30

$$V_g \left(y_+, \frac{y}{\delta} \right) = S_A(y_+) [E_1 - A_1 \ln(y_+)] + S_V(y_+) V(y_+) \left[1 - \frac{y}{\delta} \right]$$

that the boundary conditions are satisfied, i.e., as y_+ approaches infinity the value of the correction approaches $Cy_+^{-1/2}$, and as y_+ approaches zero the value of the correction approaches $E_1 - A_1 \ln(y_+)$. The switching functions are given by the relations shown in Equation 4.31. The wake correction is given by Equation

Equation 4.31

$$\begin{aligned} S_A(y_+) &= \operatorname{erfc}(m_A y_+) \\ S_V(y_+) &= \operatorname{erf}(m_V y_+) \end{aligned}$$

4.32. Marusik, Uddin and Perry show good agreement with experimental data for $A_1=1.03$, $B_1=2.39$, $E_1=6.25$, $m_A=0.0021$, and $m_v=0.0059$.

Equation 4.32

$$W_g \left(\frac{y}{\delta} \right) = B_1 \left[\frac{y}{\delta} \right]^2 \left[3 - 2 \frac{y}{\delta} \right] - A_1 \left[\frac{y}{\delta} \right]^2 \left[1 - \frac{y}{\delta} \right] \left[1 - 2 \frac{y}{\delta} \right]$$

Most turbulence intensity models require some knowledge of both the time-averaged velocity and the gradient of the time averaged velocity in the flow field. Consequently, these models are more useful for estimating the turbulence intensity from a data set than for predicting the turbulence intensity in a flow.

Turbulence Intensity Scaling in the SNS Thermal Hydraulic Experiments

Scaling turbulence intensity to aid in the design of an experiment is difficult since some knowledge of the behavior of flow field is necessary to model the turbulent behavior. For that reason, the turbulence intensity models may be more useful as tool to aid in fine tuning the test plans as initial data becomes available than as a design tool. However, using preliminary computational fluid dynamics (CFD) calculations for the water and mercury flow fields, the turbulence intensity can be estimated using Prandtl's mixing length model.

Consider a pipe flow with a 10-cm radius that is similar the inlet channel of the SNS mercury target reference design. The average velocity and velocity gradient can be estimated from preliminary CFD calculations for the WTHL test section and the mercury target. As the flow approaches the exit of the inlet channel, the velocity gradient at a distance of 5 cm from the wall is approximately 0.36 m/s/m in the WTHL test section and 0.16 m/s/m in the mercury target. The average flow velocity is approximately 3 m/s in the WTHL calculation and 1.5 m/s in the mercury target calculation, so that Reynolds number scaling is employed. Using the experimental correlation for the mixing length reported by Schetz (shown in Equation 4.19) for

Equation 4.19

$$l_m = 0.14 R - 0.08 R \left(1 - \frac{y}{R} \right)^2 - 0.06 R \left(1 - \frac{y}{R} \right)^4$$

the pipe flow considered, the mixing length is approximately 0.012 m at a distance of 5 cm from the wall. Using the estimated velocity gradients and the calculated

mixing length, the fluctuation in the streamwise velocity can be evaluated using the mixing length formulation (Equation 4.17). For the WTHL test section, the

Equation 4.17

$$u'(y_0) = U(y_0 + l_m) - U(y_0) \approx l_m \frac{\partial U}{\partial y}$$

streamwise velocity fluctuation is approximately 0.004 m/s at a distance of 5 cm from the wall. For the SNS target flow, the streamwise velocity fluctuation is approximately 0.002 m/s at a distance of 5 cm from the wall. The streamwise turbulence intensity can be calculated by normalizing the streamwise velocity fluctuation with respect to the average streamwise velocity, as shown in Equation 4.33. The turbulence intensity at a distance of 5 cm from the wall in the inlet

Equation 4.33

$$Ti_1 = \frac{u'}{U_\infty}$$

channel of the WTHL test section is approximately 0.1. In the inlet channel of the SNS mercury target, the turbulence intensity at a distance of 5 cm from the wall is also approximately 0.1. Thus, reasonable similarity can be expected in the turbulence intensity of the WTHL test section and the SNS liquid mercury target.

Since the UTATF was not designed based on CFD simulation, the turbulence intensity in that facility could not be easily predicted prior to the construction of the facility and completion of the UTATF experiments. Using the velocity data collected in the UTATF, the turbulence intensity was calculated in the same manner as it was for the WTHL. The turbulence intensity at a distance of 6 cm (1.2 times 5 cm) from the surface of the inlet channel in the UTATF is 0.08. Thus the flow field in the UTATF is actually more turbulent than the flow field in the nominal SNS mercury target.

Turbulent Kinetic Energy

The turbulent kinetic energy of a flow, like the turbulence intensity, is derived by assuming that the instantaneous velocity is the sum of a time-averaged component and a time-varying component. As in the derivation of the turbulence intensity, the total magnitude of the time-varying component of the flow is given by Equation 4.13. Thus, the kinetic energy per unit mass of fluid associated with the time-varying component of the fluid velocity can be found as shown in Equation 4.34.

Equation 4.13

$$\overline{q^2} = \overline{u'^2} + \overline{v'^2} + \overline{w'^2}$$

Equation 4.34

$$K = \frac{1}{2} \overline{q^2}$$

Like the turbulence intensity, the turbulent kinetic energy is directly related to the level of mixing that occurs within the flow field. However, unlike the turbulence intensity, a balance condition exists for the total turbulent kinetic energy of the flow. For convenience, the Einstein summation notation is typically used, where a subscript represents a vector and summation is implied by a repeated subscript, as shown in Equation 4.35. Following this convention, the three-dimensional kinetic

Equation 4.35

$$\begin{aligned} u_i &= (u_1, u_2, u_3) = (u, v, w) \\ u_i u_i &= u_1^2 + u_2^2 + u_3^2 = u^2 + v^2 + w^2 \end{aligned}$$

energy can be expressed as shown in Equation 4.36, where ν is the kinematic

Equation 4.36

$$\begin{aligned} \frac{DK}{Dt} &= -\frac{\partial}{\partial x_i} \left[\overline{u'_i \left(\frac{1}{2} u'_j u'_j + \frac{p'}{\rho} \right)} \right] - \overline{u'_i u'_j} \frac{\partial U_j}{\partial x_i} \\ &\quad + \frac{\partial}{\partial x_i} \left[\overline{\nu u'_j \left(\frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right)} \right] - \overline{\nu \frac{\partial u'_j}{\partial x_i} \left(\frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right)} \end{aligned}$$

viscosity and p' is the time fluctuation part of the local pressure. The first term on the right hand side of the equation is the convective diffusion of turbulent kinetic energy. The second term relates to the production of turbulent kinetic energy. The third term is the work done by turbulent viscous stresses. Finally, the fourth term is turbulent viscous dissipation term. In two dimensions, the turbulent kinetic energy equation can be reduced to the more familiar form shown in Equation 4.37, where the turbulent dissipation, ε , is given by Equation 4.38 (White, 1991).

Equation 4.37

$$\bar{u} \frac{\partial K}{\partial x} + \bar{v} \frac{\partial K}{\partial y} \approx \frac{\partial}{\partial y} \left[\overline{\nu' \left(\frac{1}{2} u'_i u'_i + \frac{p'}{\rho} \right)} \right] + \frac{\tau}{\rho} \frac{\partial \bar{u}}{\partial y} - \varepsilon$$

Equation 4.38

$$\varepsilon = -\nu \overline{\frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i}}$$

As in the case of the turbulence intensity, most efforts to model the turbulent kinetic energy have focused on the turbulent boundary layers that develop as fluid moves across a flat plate or over a body in cross-flow. Furthermore, most efforts have focused on the characterization of the viscous dissipation term in the kinetic energy equation rather than the characterization of the turbulent kinetic energy itself. Therefore, the prediction of the turbulent kinetic energy of the flow usually requires the use of a three-dimensional CFD code or an experiment to determine the necessary characteristics of the flow field.

Turbulent Kinetic Energy Scaling in the SNS Thermal Hydraulic Experiments

In general, turbulent kinetic energy scaling for design of an experiment is difficult since a great deal must be known about the flow field to calculate the turbulent kinetic energy directly from the turbulent kinetic energy equation. However, the streamwise velocity fluctuation models used to estimate the turbulence intensity can be used to crudely estimate the turbulent kinetic energy associated with those fluctuations.

Again consider the pipe flow with a 10-cm radius that is similar to the inlet channel of the SNS mercury target reference design. Using the Prandtl mixing length model as before, the fluctuation in the streamwise velocity can be evaluated. For the WTHL test section, the streamwise velocity fluctuation is approximately 0.004 m/s at a distance of 5 cm from the wall. For the SNS target flow, the streamwise velocity fluctuation is approximately 0.002 m/s at a distance of 5 cm from the wall. Then, from the definition of the turbulent kinetic energy, the turbulent kinetic energy resulting from the streamwise velocity fluctuations in the inlets of the WTHL test section and the SNS mercury target is 0.002 J/m³ and 0.11 J/m³, respectively. Thus, while scaling employed in the design of the WTHL test section is sufficient to ensure similarity in the turbulence intensities of the two flow fields, it is not sufficient to ensure absolute similarity in the turbulent kinetic energy of the two flow fields.

As with the turbulence intensity, the turbulent kinetic energy in the UTATF must be evaluated using experimental velocity data. Using this data, the turbulent kinetic energy can be calculated in the same manner as it was for the WTHL and mercury

target using the CFD data. The turbulence intensity at a distance of 6 cm (1.2 times 5 cm) from the surface of the inlet channel in the UTATF is 0.08 J/m³.

4.2 Similarity of Body Forces

Body forces are often neglected in the evaluation of a flow field because they typically are much smaller in magnitude than the planar forces and stresses resulting from viscous interactions with the wall. However, in the development of a scaled experiment, the consideration of body forces is important because scaling may artificially enhance or decrease the significance of the body forces in the experiment. Under the right conditions, the body forces may influence the streamlines of the flow field or encourage the development of flow instabilities.

The most commonly considered body forces are those that relate to the properties of the frame of reference in which the flow is located. For most experiments, the frame of reference is the earth's surface. In this case, body forces result because the flow is accelerated by the earth's gravitational pull or because the flow field is subject to the angular momentum imparted by the rotation of the earth. However, body forces may also result from the acceleration of the flow due to the geometry of the flow field, such as the centrifugal acceleration of a flow in a curved channel.

Scaling the Importance of Gravitational Acceleration

Gravitational acceleration is a result of the inherent attraction between any two masses as described by Newton's law of universal gravitation. The earth is typically the largest mass near an earthbound experiment. Consequently, the gravitational acceleration due to all other masses surrounding the flow field is typically negligible in comparison to the gravitational acceleration toward the center of the earth. The effect of gravitational acceleration on the motion of a fluid is mathematically represented in the conservation of momentum equation. The conservation of the momentum of a fluid element is described by the well-known Navier-Stokes equation, shown in Equation 4.39. The contributions of gravitational

Equation 4.39

$$\rho \frac{D\vec{v}}{Dt} = \rho \vec{g} - \vec{\nabla} p + \frac{\partial}{\partial x_i} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + \delta_{ij} \lambda \vec{\nabla} \cdot \vec{v}$$

acceleration to the character of a flow are typically negligible except for flows with a free surface. The Froude number provides a measure of the significance

of gravitational acceleration to the flow behavior. Although the Froude number is typically derived from the boundary condition at the free surface, it can also be found in the general non-dimensional form of the conservation of momentum equation (White, 1991).

Derivation of the Froude Number

For incompressible fluids, the Navier-Stokes equation can be reduced to the form shown in Equation 4.40 (White 1991). For a two-dimensional system, the above

Equation 4.40

$$\rho \frac{D\vec{v}}{Dt} = \rho \vec{g} - \vec{\nabla} p + \vec{\nabla}^2 \vec{v}$$

vector equation can be rewritten as two scalar equations, as shown in Equation

Equation 4.41

$$\begin{aligned} x: \quad u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} &= g_x - \frac{1}{\rho} \frac{\partial p}{\partial x} + v \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right] \\ y: \quad u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} &= g_y - \frac{1}{\rho} \frac{\partial p}{\partial y} + v \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right] \end{aligned}$$

4.41. Assuming horizontal flow in the x-direction, the gravitational acceleration term appears only in the conservation of momentum equation for the y, or vertical, direction. Typically, the pressure gradient in the vertical direction within a horizontal flow is negligible, so the equation in the y-direction can be written as shown in Equation 4.42.

Equation 4.42

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = g_y + v \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right]$$

In order to develop a scaling parameter to evaluate the contribution of gravitational acceleration to the flow behavior, the conservation of momentum equation in the y-direction must be non-dimensionalized by substituting the dimensionless variables defined in Equations 4.3a-d. The resulting equation can be written as shown in Equation 4.44. For a two-dimensional flow, the conservation of mass equation is written as shown in Equation 4.45. Non-dimensionalizing the conservation of mass equation gives the result indicated in Equation 4.46. From this equation, the

Equation 4.43

$$\begin{aligned} x^* &= \frac{x}{x_{\text{ref}}} & y^* &= \frac{y}{y_{\text{ref}}} \\ \text{(a)} & & \text{(b)} & \\ u^* &= \frac{u}{U_{\infty}} & v^* &= \frac{v}{v_{\text{ref}}} \\ \text{(c)} & & \text{(d)} & \end{aligned}$$

Equation 4.44

$$\frac{U_{\infty} v_{\text{ref}}}{x_{\text{ref}}} u^* \frac{\partial v^*}{\partial x^*} + \frac{v_{\text{ref}}^2}{y_{\text{ref}}} v^* \frac{\partial v^*}{\partial y^*} = g_y + v \left(\frac{U_{\infty} y_{\text{ref}}}{x_{\text{ref}}^3} \frac{\partial^2 v^*}{\partial x^{*2}} + \frac{U_{\infty}}{x_{\text{ref}} y_{\text{ref}}} \frac{\partial^2 v^*}{\partial y^{*2}} \right)$$

Equation 4.45

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

Equation 4.46

$$\frac{U}{x_{\text{ref}}} \frac{\partial u^*}{\partial x^*} + \frac{v_{\text{ref}}}{y_{\text{ref}}} \frac{\partial v^*}{\partial y^*} = 0$$

reference velocity in the y-direction, v_{ref} , can be defined by the relation in Equation 4.47. Consequently, the conservation of momentum equation for the y-direction

Equation 4.47

$$v_{\text{ref}} = -\frac{U}{x_{\text{ref}}} y_{\text{ref}}$$

can be written in the form of Equation 4.48. Typically, the reference length in the

Equation 4.48

$$u^* \frac{\partial v^*}{\partial x^*} + v^* \frac{\partial v^*}{\partial y^*} = \frac{x_{\text{ref}}}{y_{\text{ref}}} \frac{x_{\text{ref}}}{U_{\infty}^2} g_y + v \left(\frac{1}{U_{\infty} x_{\text{ref}}} \frac{\partial^2 v^*}{\partial x^{*2}} + \frac{x_{\text{ref}}}{U_{\infty} y_{\text{ref}}^2} \frac{\partial^2 v^*}{\partial y^{*2}} \right)$$

x-direction, x_{ref} , is assumed to represent some typical length in the system, L . The reference length in the y-direction, y_{ref} , is typically taken as the thickness as the boundary layer, δ . Then the conservation of momentum equation for the y-direction becomes the relation shown in Equation 4.49.

If the Reynolds number is defined by Equation 4.50 and the Froude number is

Equation 4.49

$$u^* \frac{\partial v^*}{\partial x^*} + v^* \frac{\partial v^*}{\partial y^*} = \frac{L}{\delta} \frac{L}{U_\infty^2} g_y + \frac{v}{U_\infty L} \left(\frac{\partial^2 v^*}{\partial x^{*2}} + \frac{L^2}{\delta^2} \frac{\partial^2 v^*}{\partial y^{*2}} \right)$$

Equation 4.50

$$Re = \frac{U_\infty L}{\nu}$$

defined by Equation 4.51, substituting the Reynolds and Froude numbers into the

Equation 4.51

$$Fr = \frac{U^2}{Lg}$$

conservation of momentum equation for the y direction yields Equation 4.52. Note

Equation 4.52

$$u^* \frac{\partial v^*}{\partial x^*} + v^* \frac{\partial v^*}{\partial y^*} = \frac{L}{\delta} \frac{1}{Fr} + \frac{1}{Re} \left(\frac{\partial^2 v^*}{\partial x^{*2}} + \frac{L^2}{\delta^2} \frac{\partial^2 v^*}{\partial y^{*2}} \right)$$

that the term L/δ is typically large since δ is typically smaller than L . Consequently, for the contribution of the gravitational acceleration to the flow development to be negligible, the term $1/Fr$ must be small.

Evaluation of the Froude Number for the SNS Thermal Hydraulic Experiments

The primary objective of the WTHL experiments is the characterization of the flow field in the mercury target using pressure and velocity measurements in combination with flow visualization studies. Therefore, the behavior of the water flow in the WTHL experiments should match the flow behavior of the mercury in the SNS target as closely as possible. Although the UTATF experiments are more qualitative in nature, they should also be somewhat representative of the flow behavior in the SNS target. For these experiments, the Froude number is used as a general measure of the significance of the effects of gravitational acceleration on the flow field. Table 4.1 lists the Froude number for the nominal SNS mercury target flow and the planned WTHL experiments, using the diameter of the inlet leg as the reference dimension. Since the nominal WTHL test case matches the inlet flow rate in the experiment to the flow rate in the nominal target design, exact similarity is achieved with respect to the effects of gravitational acceleration.

Table 4.1 – Comparison of Froude numbers for SNS thermal hydraulic experiments.

	Flow Rate (l/s, per Inlet Leg)	Froude Number
SNS Mercury Target (Nominal)	11.0	1.999
WTHL (Nominal)	11.0	1.999
UTATF (Nominal)	15.0	3.096

Scaling the Importance of Coriolis Acceleration

The Coriolis effect is the flow phenomena that results from the angular momentum imparted to the fluid by the rotation of the earth. For small systems, the effect is typically negligible. For large systems, such as the atmosphere or the oceans, the effect results in the development of large-scale rotational elements, such as the high and low pressure cyclones commonly seen in weather forecasts. However, the Coriolis effect may also be important for smaller systems in regions where time scales for flow development are sufficiently long.

Coriolis Acceleration

A fluid flowing along the earth's surface is subject to Coriolis acceleration resulting from the rotation of the earth about its axis. Coriolis acceleration acts perpendicular to the earth's surface and is responsible for such large-scale effects as the rotation of the atmosphere around points of low pressure. The Coriolis acceleration component of the total acceleration acting on a fluid is given by Equation 4.53.

Equation 4.53

$$\vec{A}_c = 2\vec{\Omega} \times \vec{v}$$

In this equation, Ω is the angular velocity of the earth's surface, about 7×10^{-5} rad/s, and v is the velocity of the fluid relative to the rotating reference frame as shown in Figure 4.2 (Pedlosky, 1985).

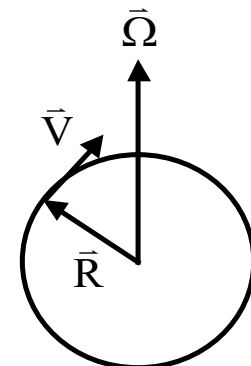


Figure 4.2 – Coriolis acceleration free body diagram.

Derivation of the Rossby and Ekman Numbers

In order to include the effects of Coriolis acceleration, the equation representing the conservation of momentum must be rewritten to describe a system that rotates with a steady angular velocity relative to the inertial frame. Thus, the equation for the

conservation of momentum can be rewritten as Equation 4.54, where $\vec{v}$ is the flow velocity vector relative to the rotating frame, $\vec{G}$ is the external force acting on the fluid element, and s is the distance of the point $\vec{r}$ from the rotation axis. If $\vec{G}$

Equation 4.54

$$\rho \left[\frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \vec{\nabla}) \vec{v} \right] + 2\rho (\vec{\Omega} \times \vec{v}) = \rho \vec{G} - \vec{\nabla} p + \rho \vec{\nabla} \left(\frac{1}{2} \Omega^2 s^2 \right) + \mu \vec{\nabla}^2 \vec{v}$$

is a conservative force field that can be described by Equation 4.55, where V is

Equation 4.55

$$\vec{G} = -\vec{\nabla} V$$

a potential function, then the momentum equation can be rewritten in the form of Equation 4.56 where the pressure term is modified as shown in Equation 4.57.

Equation 4.56

$$\frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \vec{\nabla}) \vec{v} + 2 \vec{\Omega} \times \vec{v} = -\vec{\nabla} \bar{p} + \nu \nabla^2 \vec{v}$$

Equation 4.57

$$\bar{p} = \frac{p}{\rho} + V - \frac{1}{2} \Omega^2 s^2$$

In order to develop a scaling parameter to evaluate the significance of Coriolis acceleration to the development of the flow field, the vorticity equation must be non-dimensionalized using the transformations defined in Equations 4.58a-e. The

Equation 4.58

$$\begin{aligned} \vec{r}^* &= \frac{\vec{r}}{L} & t^* &= \frac{t}{T} & \vec{\Omega} &= \hat{k} \bar{\Omega} \\ (a) & & (b) & & (c) & \\ \vec{v}^* &= \frac{\vec{v}}{U_\infty} & p^* &= \frac{p}{\rho \Omega L U_\infty} \\ (d) & & (e) & & & \end{aligned}$$

momentum equation then takes the form of Equation 4.59. Noting that the Rossby

Equation 4.59

$$\frac{1}{\Omega T} \frac{\partial \vec{v}^*}{\partial t} + \frac{U_\infty}{L \Omega} (\vec{v}^* \cdot \vec{\nabla}) \vec{v}^* + 2 \hat{k} \times \vec{v}^* = -\vec{\nabla} p^* + \frac{\nu}{L^2 \Omega} \nabla^2 \vec{v}^*$$

number is defined by Equation 4.60 and the Ekman number is defined by Equation

Equation 4.60

$$Ro \equiv \frac{U_\infty}{L \Omega}$$

4.61, the momentum equation can be rewritten as Equation 4.62.

Equation 4.61

$$E \equiv \frac{\nu}{L^2 \Omega}$$

Equation 4.62

$$\frac{1}{\Omega T} \frac{\partial \vec{v}^*}{\partial t} + Ro (\vec{v}^* \cdot \vec{\nabla}) \vec{v}^* + 2 \hat{k} \times \vec{v}^* = -\vec{\nabla} p^* + E \nabla^2 \vec{v}^*$$

The Rossby number is of the order of the ratio of the inertial forces to the Coriolis forces, and the Ekman number is of the order of the ratio of the viscous forces to the Coriolis forces. Thus, when both numbers are small (i.e., less than one) the Coriolis force will significantly affect the development of the flow field. Otherwise, the force is negligible in comparison to the other forces acting on the fluid (Roberts, 1978).

Evaluation of the Rossby and Ekman Numbers for the WTHL

The Rossby and Ekman numbers are used to evaluate the effects of the Coriolis force on the development of the flow field in the SNS mercury target as well as the WTHL and UTATF experiments. The Rossby and Ekman numbers for the SNS thermal hydraulic experiments and the nominal SNS mercury target are shown in Table 4.2. While the Ekman numbers are small enough to indicate that the Coriolis force may not be negligible with respect to the viscous force in the SNS mercury target, the Rossby numbers indicate that the Coriolis force is indeed negligible in all of the systems because the inertia of the fluid is large. Thus, the Rossby number

Table 4.2 – Rossby and Ekman numbers for SNS thermal hydraulic experiments.

	Flow Rate (l/s, per Inlet Leg)	Ekman Number	Rossby Number
SNS Mercury Target (Nominal)	11.0	0.138	1.00E+05
WTHL (Nominal)	11.0	1.440	1.00E+05
UTATF (Nominal)	15.0	11.40	1.14E+05

is the critical parameter for the evaluation the importance of the Coriolis force in the SNS target. The Rossby number for the nominal WTHL tests is identical to the Rossby number of the nominal SNS mercury target.

Scaling the Importance of Centrifugal Acceleration

Centrifugal acceleration results when a fluid flows through a curved channel or along a curved streamline. For any flow that follows a curved streamline, centrifugal acceleration can exert some influence over the flow patterns and pressure distributions within the flow field. However, these effects are typically negligible for high Reynolds number flows. For the special case of flows in a narrow curved channel, centrifugal acceleration induces a pressure gradient across the flow field between the concave and convex walls. The pressure gradient often results in the development of flow instabilities along one or both of the channel walls.

Development of Centrifugal Flow Instabilities

The effects of centrifugal acceleration are explored in several classical flow problems that were examined in the late 19th and early 20th centuries. The first is the case of Couette flow in the annulus between two rotating cylinders, commonly known as the Taylor problem. In the Taylor problem, vortices develop in the gap between the two cylinders, as shown in Figure 4.3, for certain values of the angular velocities of the two cylinders and the gap width between them. The Taylor number, used to quantify the likelihood of the development of Taylor vortices in the flow, is given by Equation 4.63, where r_i is the radius of the inner cylinder, r_o is the radius of the outer cylinder, ν is the viscosity of the fluid, Ω_i is the angular velocity of the inner cylinder, and Ω_o is the angular velocity of the outer cylinder. For

large values of the Taylor number, the instability will appear. For very large values, full turbulent flow develops and the instability may disappear completely (White, 1991).

Equation 4.63

$$Ta = \frac{r_i (r_o - r_i)^3 (\Omega_i^2 - \Omega_o^2)}{\nu^2}$$

The second classical flow problem that examines the effects of centrifugal acceleration on the development of a flow field is the Dean problem. The Dean problem focuses on the development of Taylor-like instabilities in a narrow curved channel. The likelihood of the development of the Dean instability can be evaluated using the Dean number, given by Equation 4.64, where v is the flow velocity, L is the characteristic length, ν is the kinematic viscosity, and r_c is the radius of curvature of the flow. As in the Taylor problem, the instability appears when the Dean number is large. When the Dean number is very large, full turbulence begins to development and the effects of the instability may disappear entirely (White, 1991).

Equation 4.64

$$De = \frac{vL}{\nu} \left(\frac{1}{2 r_c} \right)^{\frac{1}{2}}$$

The third classical problem that examines the effects of centrifugal acceleration is the Goertler problem. The Goertler problem is really an extension of the Dean problem. The Goertler problem removes the narrow gap width requirement of the Dean problem by focusing on the boundary layer along a concave wall. Goertler observed the development of a Taylor-like instability within the boundary layer as shown in Figure 4.4. The Goertler number, which evaluates the likelihood of the development of Goertler vortices in a boundary, is shown in Equation 4.65, where δ_r is the boundary layer thickness. The boundary layer thickness along a concave surface is given by Equation 4.66, where $\bar{x}$ is the distance from the

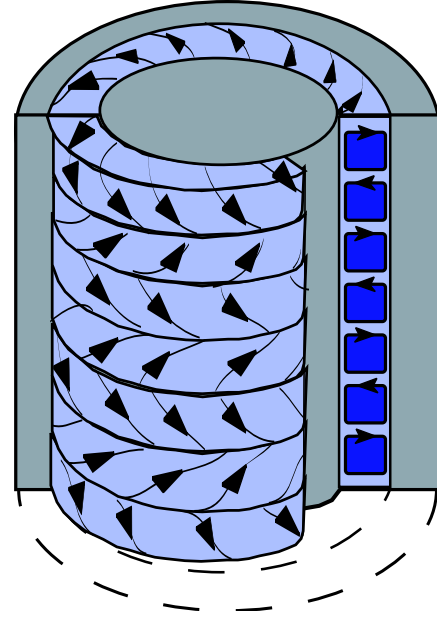


Figure 4.3 – Development of Taylor vortices in an annular region between rotating cylinders.

Equation 4.65

$$Go = \frac{v\delta_r}{\nu} \left(\frac{\delta_r}{r_c} \right)^{\frac{1}{2}}$$

Equation 4.66

$$\delta_r = \left(\frac{\nu \bar{x}}{v} \right)^{\frac{1}{2}}$$

leading edge of the concave surface (Floryan, 1991). The Goertler instability first appears for values of the Goertler number of about 0.6 (Duncan, Thom and Young, 1970). For large values of the Goertler number, the flow is typically fully turbulent. The instability may continue to appear even for very turbulent flows because the instability is contained within the boundary layer. However, the steady vortex lines, like those seen in Figure 4.4, begin to meander across the surface as the flow transitions to turbulence (Floryan, 1991).

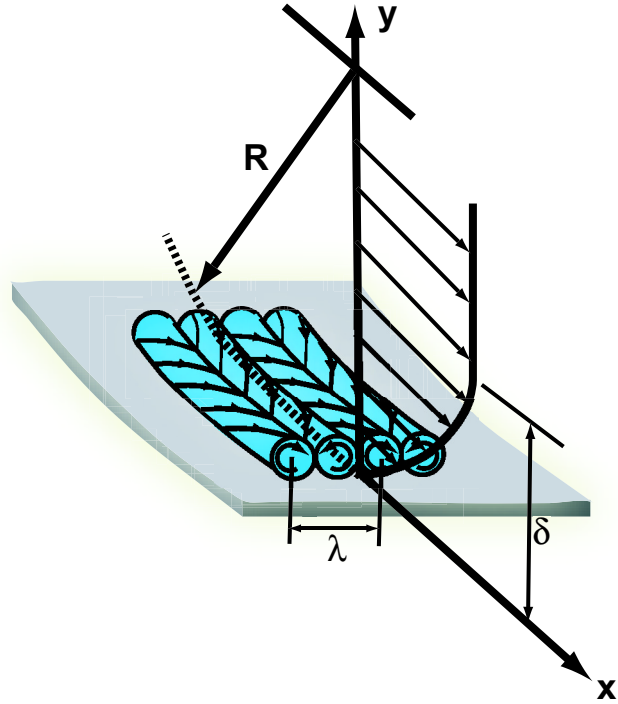


Figure 4.4 – Development of Goertler vortices along a concave surface.

Goertler-like instabilities that result from centrifugal acceleration may develop along any curved surface over which fluid flows. Typically, the Goertler instability results in a slight enhancement of heat transfer and a slight increase in head losses in the channel. In laminar flows, hot stripes may appear in the downwash region between two vortex streams. But in turbulent flows, the vortex streams are typically not stationary long enough for hot stripes to develop (Crane and Sabzvari, 1989).

Evaluation of Centrifugal Acceleration in the SNS Thermal Hydraulic Experiments

The Goertler number provides a measure of the likelihood that centrifugal effects are important in a curved flow region. The Goertler number in the portion of the flow that turns around the flow baffle in the SNS mercury target is approximately 28.9. The Goertler numbers in the WTHL and UTATF experiments at the same position are 16.1 and 9.93, respectively. Since the Goertler number is much

greater than 1, the effects of centrifugal acceleration are probably not significant to the behavior of the bulk flow field. A large Goertler number simply indicates that other phenomena are more significant in the development of the characteristics of the bulk flow field and does not preclude the development of Goertler or other similar vortex structures in the boundary layer. Therefore, the portion of the experimental program that examines heat transfer should consider the possibility of such structures existing in both the bulk flow region and the separate beam window cooling jacket and anticipate their effects on such issues as hot striping and mercury non-wetting.

4.3 Similarity of Forces Resulting from Energy Deposition

As a result of the proton interactions with the liquid mercury and the target structure, the proton beam deposits approximately 2 MW of power in the SNS mercury target. The power is deposited in equal pulsed doses at a frequency of 60 Hz. On a time-averaged basis, the energy deposition results in spatial gradients in local temperatures and fluid properties. On a time-dependent basis, the pulsed nature of the proton beam leads to the rapid fluctuations in the energy deposition. Consequently, the fluid undergoes rapid changes in local temperature and pressure as the conservation of energy equations are stringently obeyed. The rapid expansion of the fluid results in a phenomenon known as thermal shock.

Scaling the Effects of Spatial Temperature Gradients

As a result of the energy deposition, thermal gradients will develop in the liquid mercury flow field. Thermal gradients incite the development of density gradients in the fluid as a result of the response of the properties of the fluid to the changes in the local temperature. Buoyancy-driven flows often appear in regions with substantial thermal gradients and, indeed, buoyancy is the primary driving force in natural convection systems. However, the effects of buoyancy are typically negligible in forced flow environments. Yet, in the development of a scaled experimental program, it is important to evaluate the importance of phenomena in the system to be modeled that cannot be captured in the planned experiments.

Scaling the Effects of Buoyancy Forces

Scaling the effects of buoyancy on a flow field may be approached in two ways. In the first case, the fluid flow is assumed to be entirely within a boundary layer, laminar, steady, and incompressible with the buoyancy effects included in the momentum balance as a body force. The relationship between the buoyancy

force and the temperature gradient is simplified through the introduction of the volumetric thermal expansion coefficient. In the second case, the Boussinesq approximate form of the conservation equations is used to consider the effects of the temperature gradient more directly.

Derivation of the Grashof Number

The development of the first method of evaluating the significance of the buoyancy force is derived from the equations governing a laminar boundary layer flow in the direction opposite the acceleration of gravity. The flow is assumed to be steady, two dimensional, constant-property, and incompressible. Subject to these constraints, the appropriate form of the momentum equation is shown in Equation 4.67, where x is the direction opposite the acceleration of gravity. Noting

Equation 4.67

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} - g + \nu \frac{\partial^2 u}{\partial y^2}$$

that if there is no body force in the y -direction, the pressure gradient in that direction must be zero. Therefore, the pressure gradient in the x -direction at any point in the boundary layer must equal the pressure gradient in the x -direction at any point outside the boundary layer where both velocity components are assumed to be zero. Then Equation 4.67 can be rewritten in the form of Equation 4.68.

Equation 4.68

$$\frac{\partial p}{\partial x} = -\rho_{\infty} g$$

The substitution of this reduced form back into the original equation results in the expression shown in Equation 4.69, which must apply everywhere in the boundary layer being considered.

Equation 4.69

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \frac{g}{\rho} (\rho_{\infty} - \rho) + \nu \frac{\partial^2 u}{\partial y^2}$$

The first term on the right hand side of Equation 4.69 represents the buoyancy force that drives the natural convection flow. This force is derived from the variation in the density that results from the presence of the temperature gradient. The origin of the variation is made more explicit in the approximate form of the

momentum equation through the introduction of the volumetric thermal expansion coefficient, shown in Equation 4.70. Using this thermodynamic property, the

Equation 4.70

$$\beta = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p$$

momentum equation becomes the form shown in Equation 4.71. A convenient

Equation 4.71

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = g\beta (T - T_\infty) + \nu \frac{\partial^2 u}{\partial y^2}$$

benefit of the assumptions inherent in this development is that the buoyancy effects are confined to the momentum equation, and the other conservation equations remain unchanged.

As in previous discussions, it is convenient to consider the non-dimensionalized form of the conservation equations governing the flow field. For this case, the appropriate dimensionless variables, shown in Equations 4.72a-e, are substituted

Equation 4.72

$$x^* \equiv \frac{x}{L} \quad y^* \equiv \frac{y}{L}$$

(a) (b)

$$u^* \equiv \frac{u}{U_\infty} \quad v^* \equiv \frac{v}{U_\infty} \quad T^* \equiv \frac{T - T_\infty}{T_s - T_\infty}$$

(c) (d) (e)

in the form of the momentum equation shown in Equation 4.71 to get the non-dimensionalized form shown in Equation 4.73. The dimensionless number that

Equation 4.73

$$u^* \frac{\partial u^*}{\partial x^*} + v^* \frac{\partial u^*}{\partial y^*} = \frac{g\beta (T_s - T_\infty)L}{U_\infty^2} T^* + \frac{1}{\text{Re}} \frac{\partial^2 u^*}{\partial y^{*2}}$$

results from the buoyancy force is known as the Grashof number. This form of the number is inconvenient because the reference velocity is unknown. Therefore a more convenient form, shown in Equation 4.74, is commonly used where the

Equation 4.74

$$Gr = \frac{g \beta (T_s - T_\infty) L}{U_\infty^2} \left(\frac{U_\infty L}{\nu} \right)^2 = \frac{g \beta (T_s - T_\infty) L^3}{\nu^2}$$

original form has been multiplied by the square of the Reynolds number.

In buoyancy driven flow, the Grashof number is analogous to the Reynolds number in forced convective flow. The Reynolds number is a measure of the ratio of the inertial forces acting on a fluid element to the viscous forces acting on that element. Similarly, the Grashof number is a measure of the ratio of the buoyancy forces acting on a fluid element to the viscous forces acting on the element. When a forced convective flow is superimposed on a buoyancy driven flow, the relative importance of the two flow fields can be evaluated using the ratio of the Grashof number to the square of the Reynolds number, Gr/Re^2 . When the ratio is much less than one, the forced convection effects dominate the natural buoyancy effects. When the ratio is much greater than one, the buoyancy effects dominate. When the ratio is of order 1, the combined effects of the two superimposed flow fields must be considered (Incropera and Dewitt, 1990).

Evaluation of the Grashof Number for the SNS Thermal Hydraulic Experiments

Since the pulsed volumetric energy deposition in the SNS mercury target cannot be mimicked in SNS thermal hydraulic experiments in a practical manner, scaling of the buoyancy force cannot be used to develop those experiments. However, it is important to establish the significance of the effects of buoyancy forces on the development of the flow field in the liquid mercury target. For the bulk mercury flow of the SNS target, the maximum Grashof number is 1.90×10^{10} , and the ratio of the Grashof number to the square of the Reynolds number, Gr/Re^2 , is 9.05×10^{-3} . Thus, the forced convective component of the flow field dominates and the buoyant force is negligible. This implies that the inability to reproduce the buoyant forces found in the SNS mercury target in the SNS thermal hydraulic experiments does not hamper the ability of those experiments to provide reliable data about the character of the flow field.

Derivation of the Richardson Number

The second method of evaluating the importance of the buoyancy force in the development of a flow field is derived from the Boussinesq approximation to the equations of motion, which define the fields of velocity and temperature as shown

in Equations 4.75a-c. For convenience, the summation notation is again used

Equation 4.75

$$\frac{\partial u_k}{\partial x_k} = 0 \quad (a)$$

$$\frac{\partial u_i}{\partial t} + u_k \frac{\partial u_i}{\partial x_k} = -\frac{\partial p}{\partial x_i} - \frac{\rho - \rho_a}{\rho_a} g_i + \mu \nabla^2 u_i \quad (b)$$

$$\frac{\partial T}{\partial t} + u_k \frac{\partial T}{\partial x_k} = k \nabla^2 T \quad (c)$$

where repeated subscripts are used to indicate summation of components. If these equations are put in their non-dimensional form using the dimensionless quantities shown in Equations 4.76a-c, then three constant coefficients appear in

Equation 4.76

$$x_i^* = \frac{x_i}{L} \quad T^* = \frac{\theta}{\theta_0} \quad (a) \quad (b)$$

$$u_i^* = \frac{u_i}{U_\infty} \quad (c)$$

the three equations that are combinations of the three independent parameters. The constant coefficients are the Reynolds number, the Prandtl number, and the Richardson number. The Reynolds number has been discussed at length in previous sections. The Prandtl number, shown in Equation 4.77, is the ratio

Equation 4.77

$$Pr = \frac{\rho c_p \nu}{k}$$

of the momentum and thermal diffusivities. The Prandtl number indicates the relationship between the development of the thermal and viscous boundary layers. The Richardson number, shown in Equation 4.78, is essentially the ratio of the buoyancy forces to the inertial forces. This form of the Richardson number,

Equation 4.78

$$Ri = \frac{g L \theta_0}{T_\infty U_\infty^2}$$

often called the overall Richardson number, is applicable the entire flow field. Other forms of the Richardson number are more closely tied to a specific position within the flow. Interestingly, the overall Richardson number is identical to the Grashof number in the special case of an ideal gas flow. In ideal gasses, the thermal expansion coefficient is equivalent to the reciprocal of the gas temperature (Townsend, 1976).

Evaluation of the Richardson Number for the SNS Thermal Hydraulic Experiments

Since the SNS thermal hydraulic experiments are not heated systems, the Richardson number is not applicable to those flow fields. However, it can be used to examine the importance of the buoyant forces in the development of the SNS target's bulk mercury flow. The Richardson number for the SNS target is 0.142. Since the Richardson number is less than 1, the inertial forces dominate and the buoyant forces have a minimal effect on the development of the flow field. Therefore, the absence of these forces in the scaled experiments do not limit their ability to simulate the flow behavior of the bulk mercury flow.

Evaluation of the Effects of Thermal Shock in the SNS Mercury Target

Thermal shock is a complex acoustics problem that is extremely difficult to simulate. The primary concern in the SNS target is that pressure gradients resulting from the pulsed energy deposition will significantly reduce the lifetime of the target structure. A secondary concern is the development of secondary flow patterns within the target's mercury flow systems as a result of these pressure oscillations – essentially the target structure could behave like a pump. In order to bound the impact of the thermal shock on the flow field, a simplified problem was evaluated.

Consider the closed pipe shown in Figure 4.5. The maximum displacement of the thin steel sheet as a result of the induced pressure wave can be estimated if the pressure disturbance is treated as a resonance in this closed pipe. The peak magnitude of the pressure wave is about 30 MPa and occurs almost immediately after the proton pulse enters the target. A rough estimate can

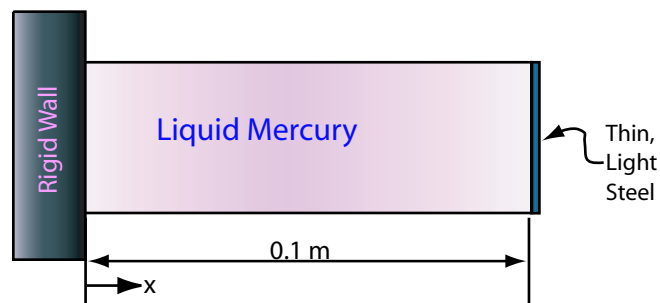


Figure 4.5 – Closed pipe geometry used for bounding evaluation of thermal shock effects.

be obtained and the problem is greatly simplified if absorption dampening is neglected.

With the assumptions stated above, the pressure at any space-time coordinate within the liquid mercury can be represented by Equation 4.79, where the incident

Equation 4.79

$$p = A \cos (\omega t + k x) + B \cos (\omega t - k x)$$

and reflective waves can be separated as shown in Equation 4.80. If the

Equation 4.80

$$\begin{aligned} p_i &= A \cos (\omega t + k x) \\ p_r &= B \cos (\omega t - k x) \end{aligned}$$

coefficients of Equation 4.79 are known, the displacement of the liquid mercury resulting from the pressure field can be found using Equation 4.81. Furthermore,

Equation 4.81

$$\xi = \frac{A}{\rho_o c \omega} \sin (\omega t - k x) + \frac{B}{\rho_o c \omega} \sin (\omega t + k x)$$

the volumetric fluid velocities resulting from the initial and reflected pressure waves can be found using Equation 4.82. The acoustic impedance at any point in the pipe

Equation 4.82

$$\begin{aligned} u_i &= \frac{p_i}{\rho_o c / A_{xs}} \\ u_r &= \frac{p_r}{\rho_o c / A_{xs}} \end{aligned}$$

is given by Equation 4.83 (Kinsler and Frey, 1962).

Equation 4.83

$$Z = \frac{p_i + p_r}{u_i + u_r} = \frac{\rho_o c}{A_{xs}} \frac{p_i + p_r}{p_i - p_r}$$

If the point at which the mercury contacts the rigid wall, $x = 0$, is considered, the impedance can be rewritten as shown in Equation 4.84. Recognizing that the

Equation 4.84

$$z_0 = \frac{\rho_0 c}{A_{xs}} \frac{A + B}{A - B}$$

impedance at this point must be infinite, the relationship between the coefficients can be determined as shown in Equation 4.85. Taking the limit of this relation

Equation 4.85

$$A = \lim_{z_0 \rightarrow \infty} \frac{\left(\frac{\rho_0 c}{A_{xs}} + z_0 \right)}{\left(z_0 - \frac{\rho_0 c}{A_{xs}} \right)} B$$

shows that the coefficients must be equal, which is a direct consequence of the assumption of no absorption dampening. This allows the impedance equation to be simplified to the form shown in Equation 4.86.

Equation 4.86

$$z = - \frac{\cos \omega t \cos kx}{\sin \omega t \sin kx}$$

At the point at which the mercury contacts the thin steel sheet, $x = 0.1$ m, the impedance is zero. Therefore, Equation 4.86 can be used to determine the time at which the maximum displacement occurs at the point $x = 0.1$ m. Using the appropriate fluid properties, the maximum displacement occurs 0.68 ms after the introduction of the pressure disturbance at $x = 0$. Using Equation 4.81, the maximum displacement is calculated to be 1.3 mm. The average velocity of the thin steel sheet is 1.9 m/s and the average acceleration is 2800 m/s².

Chapter 5 Simulation of the SNS Target Bulk Mercury Flow

The thermal hydraulic characteristics of the initial SNS mercury target and all subsequent redesigns will be evaluated using a Computational Fluid Dynamics (CFD) simulation. The use of CFD simulation for design is dramatically less expensive than designing by experiment. The length of time needed for development is also typically reduced. However the CFD software used to create the simulation must be well qualified for the types of flow patterns expected in the flow field.

5.1 Description of the CFD Model

The general purpose CFD code CFX4.2 is used to provide thermal hydraulic evaluations of the SNS mercury target. The software uses a finite volume pressure correction method to simulate the flow field. The CFD simulation provides predictions of the temperature and velocity profiles within the target.

The SNS target model is a fully three-dimensional model of the bulk mercury flow in the target vessel, significant portions of the inlet piping upstream of the target, and the single outlet duct. For evaluation of the velocity profiles, the computational domain typically consists entirely of liquid mercury. However, the solid components of the target may be included if temperature distributions in those regions are desired. Two planes of geometric symmetry exist in the target structure, allowing the use of two-dimensional simulation when applicable. The full three-dimensional simulation is preferred, however, since the flow field exhibits some asymmetric tendencies. Figure 5.1 shows the geometry included in the full three-dimensional model (Wendel, 2001).

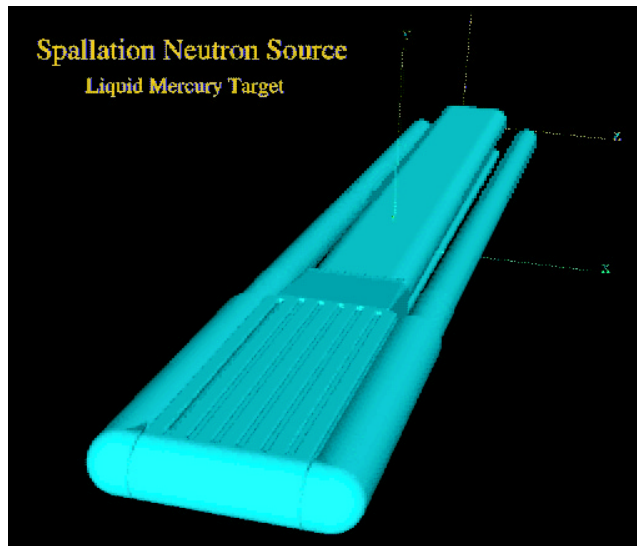


Figure 5.1 – Target geometry included in the three-dimensional CFD model. To see additional views of the geometry, click on the image to activate the animation (Wendel, 2001 – Used with permission).

A uniform inlet velocity based on the nominal flow conditions is specified at the inlet boundary since an adequate length of the inlet is included for velocity profile development. The outlet boundary condition is specified as a uniform pressure. The inlet temperature is specified as 60 °C and the outlet temperature is specified as 90 °C. The enthalpy rise in the target is 1.2 MW, or 60 percent of the beam power (Wendel, 2000).

The discretized geometry of the three-dimensional model includes 2 to 3 million computational cells. The three-dimensional model requires long simulation times – up to 3 weeks of CPU time on a 500 MHz Dec ALPHA Personal Workstation – to become well converged in a steady state calculation. Simplified two-dimensional calculations may converge in just a few hours. A sample of the three-dimensional discretization is shown in Figure 5.2 (Wendel, 2000).

The numerical model used to evaluate turbulent transport within the system can have a dramatic impact on the simulation results. The model used in the simulation of the SNS mercury target is the Reynolds-Averaged Navier Stokes (RANS) model, which assumes an eddy viscosity calculated with the RNG $k-\epsilon$ turbulence model. The boundaries of the computational domain that represent the interface between the fluid and the surrounding structure are modeled as no-slip

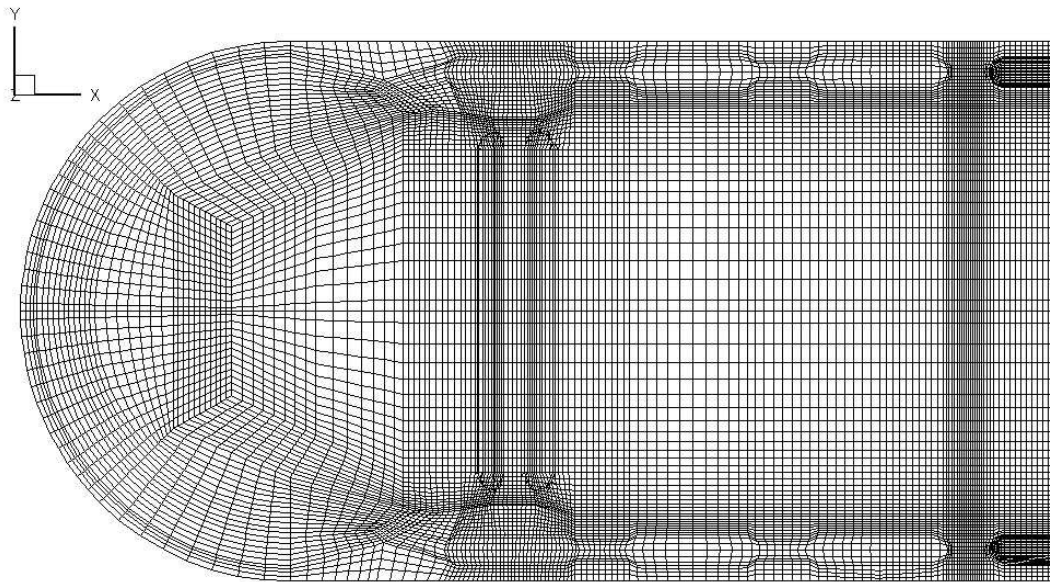


Figure 5.2 – Close up view of the discretization of the SNS mercury target (Wendel, 2000 – Used with permission).

walls. The standard law-of-the-wall approximation is used to impose the shear stress boundary condition. The use of low Reynolds number turbulence models which do not use wall functions to model the near wall regions is not practical because of the additional discretization required to resolve the viscous sublayer. Direct numerical simulation and large eddy simulation of the bulk flow regions are also not practical.

5.2 Predicted Flow Characteristics in the SNS Mercury Target

While the focus of the CFD studies is on simulating the flow in order to evaluate temperature profiles, the components of the CFD simulation of primary interest to this study are those that focus on the velocity profiles within the target. The initial velocity profile calculations consider only half of the SNS target, dividing the target at the geometric center along the horizontal axis (i.e., the vertical midplane). Assuming the flow field exhibits the same symmetry as the target reduces the number of computational volumes needed and, consequently, the time required for the calculation to converge. The streamlines predicted by these preliminary studies are shown in Figure 5.3.

Taking advantage of the geometric symmetry of the SNS target forces the simulation to assume the flow is symmetric

as well. In order to evaluate the symmetry of the flow field across the vertical midplane, the target was modeled in two dimensions assuming the geometry of the horizontal midplane determines the primary flow patterns. Interestingly, these steady state calculations result in three separate steady solutions – one symmetric and two asymmetric. Indeed, if long calculation times are allowed, the steady state simulation may migrate from one steady state solution to another without

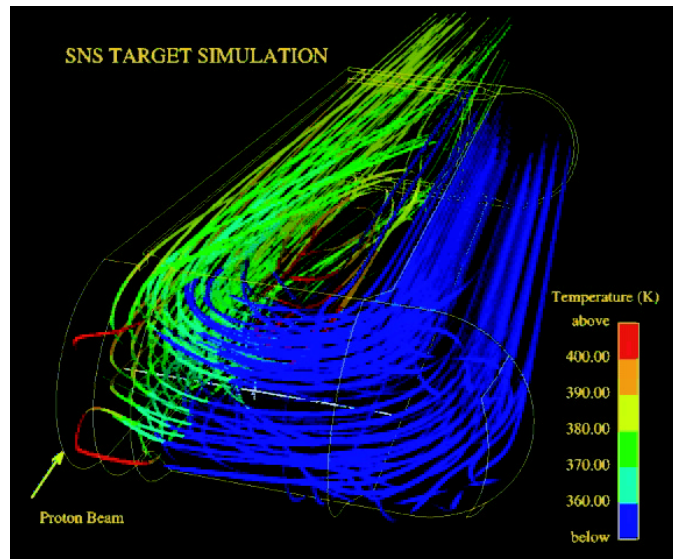


Figure 5.3 – Streamlines in the SNS mercury target with the assumption of horizontal symmetry. In this image, the color of the streamline indicates the temperature of the fluid at that location. To see simulated particles moving through the model, click on the image to activate the animation (Wendel, 2001 – Used with permission).

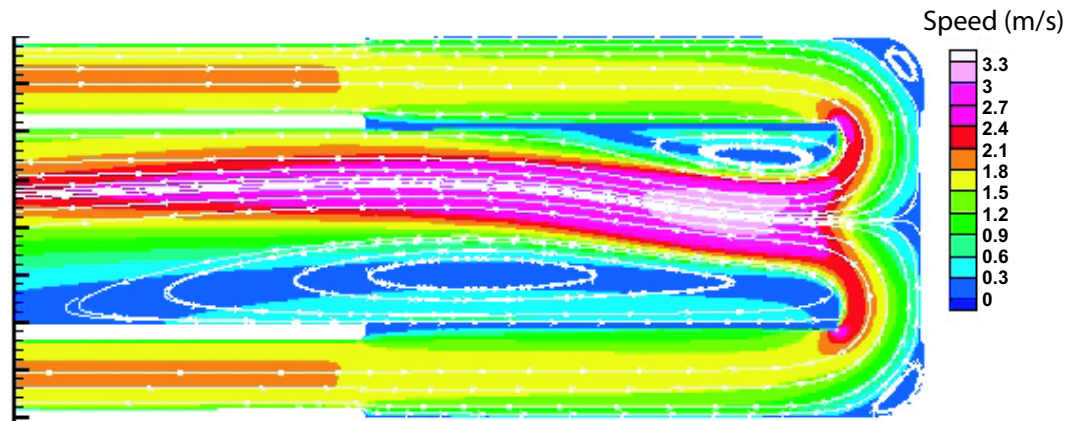


Figure 5.4 – Velocity profiles from the steady state two-dimensional simulations of the SNS mercury target. Shown is one of the two asymmetric solutions. The other asymmetric solution is simply a mirror image of the steady state solution (Wendel, 2001 – Used with permission).

deliberate external perturbation. Transient simulations do not exhibit the same ability to predict multiple steady solutions without perturbation. An example asymmetric result from this simulation is shown in Figure 5.4. The other asymmetric result is simply a mirror image of the result shown. The symmetric result is identical to the result seen in Figure 5.3.

Upon discovery of this asymmetry, the target model was modified so that the upper half of the target was modeled in three dimensions. This modification is necessary to examine the impact of the vertical curvature of the target design on the results seen in the two-dimensional studies. Again, the simulation is a steady state calculation and multiple steady solutions may be obtained. One solution is shown in Figure 5.5.

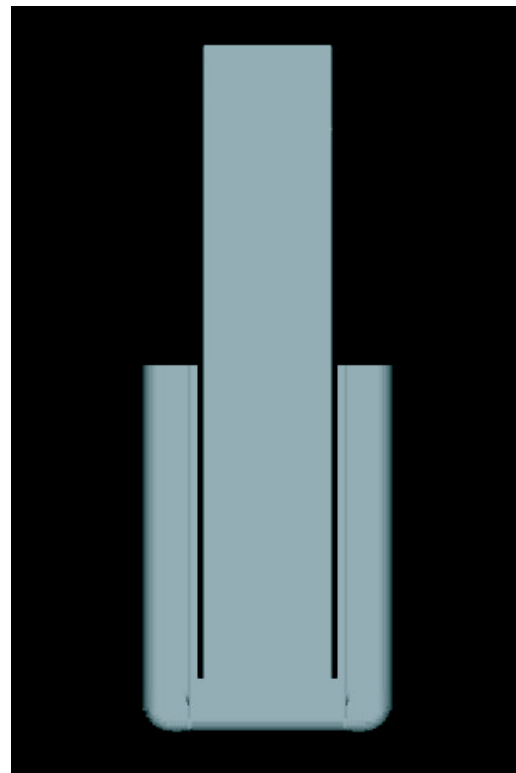


Figure 5.5 – Simulation of the SNS target flow assuming vertical symmetry. Click on the image to activate the animation. The color of the streamline indicates the magnitude of the velocity (Wendel, 2001 – Used with permission)

The appearance of multiple solutions in the steady state simulations indicates that there are likely multiple modes of behavior in the SNS target flow. The switching from one mode to another in the two-dimensional steady state calculations does not indicate the steady state simulation predicts periodic swapping between modes in the flow field. Rather it indicates that the numerical algorithm allows the simulation to move between all possible steady solutions.

The evaluation of temperature distributions in the SNS mercury target based on these simulations indicate that there are three regions in the bulk mercury flow that dramatically effect the temperature distribution in the target structure. The first region is the recirculation zone that is predicted in the nose region of the target near the center of the beam window, which could cause the development of a hot spot in this region if the recirculation zone is stable. The other two points of concern occur on the exit channel side of the flow baffles near the center of each primary recirculation zone. Similarly, these recirculation zones could lead to the development of hot spots on the flow baffles if the typical residence time of a fluid element in the recirculation zone is long enough to allow the fluid element to be exposed to multiple proton beam pulses.

5.3 Review of Previous Studies

Any numerical simulation is only as good as the mathematical model on which it is built. While there are no experimental data sets in the literature that could be used for direct validation of the CFD model, there are extensive studies discussing significant components of the model. These studies provide a basis for the assumptions made in the development of the CFD model of the SNS mercury target and allow additional benchmarking studies of the CFD model.

Because the 180-degree turn is commonly found in many engineering applications from heat exchangers to chemical reactors, this geometry is frequently considered in fluid dynamics literature. In 1986, Meztger and Sahm published a study investigating heat transfer in 180-degree turn geometries. This study examined the impact of small changes in the flow geometry, specifically in the ratio of the inlet flow area to the outlet flow area, on the average heat-transfer coefficient in the test section. They observed that very small variations in the ratio of the flow areas lead to substantial variations in the heat-transfer coefficient. In general, as the inlet flow area increases with respect to the outlet flow area additional heat transfer is observed.

A number of researches, including, Han, Chandra and Lau (1988) as well as Hirota, et al. (1999) have demonstrated that steep gradients in the heat-transfer coefficient develop as the fluid makes the turn and moves through the outlet channel. The development of the gradients is observed to be a consequence of the development of a separation bubble and recirculation zone near the leading edge of the flow baffle and the subsequent reattachment of the flow downstream. The heat-transfer coefficient is dramatically increased in the recirculation zone.

The next step beyond experimental identification of the flow behavior is the evaluation of the applicability of existing flow models for the prediction of the observed flow behaviors. Heyerichs and Pollard (1996) investigated the applicability of the $k-\epsilon$ model for the prediction of turbulent behavior in separation regions of turbulent flow fields, specifically considering the backward facing step problem. Their study concludes that the $k-\epsilon$ model provides reasonable agreement with experimental data in the developed near-wall flow regions, but do not perform as well in the recirculation zone or in the reattachment region downstream of the recirculation zone.

Liou, Hwang, and Chen (1992) examine the predictive capabilities of the $k-\epsilon$ model and the Algebraic Reynolds Stress model. Their study uses velocity data collected by LDV in a complex geometry that includes two impinging turbulent inlet flows and substantial stationary recirculation zones in the outlet flow. The study concludes that the $k-\epsilon$ model and the Reynolds stress model both provide reasonable agreement with experimental observation. The Reynolds stress model provides better agreement in the regions where complex flow patterns develop, but these calculations are much more computationally intensive.

While these studies provide a basis for the assumptions inherent in the computational model, they provide little insight into the expected flow behavior in the SNS target geometry. One study, published by Khanzhankov in 1964, considers flow geometries similar to that of the University of Tennessee Air Test Facility (UTATF) test section. The study focuses on experimental measurement of pressure losses in the test section and evaluation of the effects of modifications to the geometry on the total pressure drop. Modifications evaluated in the study include the addition of turning vanes around the leading edge of the flow baffles and the addition of a flow divider plate in the region of the exit channel where the two inlet flows begin to mix. The study indicates that the most effective means of

reducing the total pressure drop in the system is the addition of a divider plate in the exit channel.

Chapter 6 Flow Visualization Experiments to Characterize Flow Patterns in the SNS Mercury Target

Flow visualization techniques provide the opportunity to observe the behavior of a flow field directly so those regions of interest for further study may be identified. The results of such studies are more qualitative than quantitative but can be useful in prioritizing more quantitative measurements and identifying unexpected phenomena. These studies are also useful for verification of the general flow behavior predicted by Computational Fluid Dynamics (CFD) simulation. Flow visualization is accomplished in the University of Tennessee Air Test Facility (UTATF) using smoke injection and thermochromic imaging. In the Water Thermal Hydraulic Loop (WTHL) test section flow visualization is accomplished using bubble injection and dye injection.

6.1 Flow Visualization in the UTATF

Flow visualization experiments in the UTATF were the first experiments to examine the flow behavior in the SNS liquid mercury target. Initially, tufts of yarn pinned in the test section were used to characterize the flow of air through the UTATF test section. One of these preliminary studies is shown in Figure 6.1. These studies indicate that some large scale turbulent elements may appear in the exit channel flow field. In order to provide more resolution in the later UTATF flow visualization studies, a combination of smoke injection and thermochromic imaging were used.

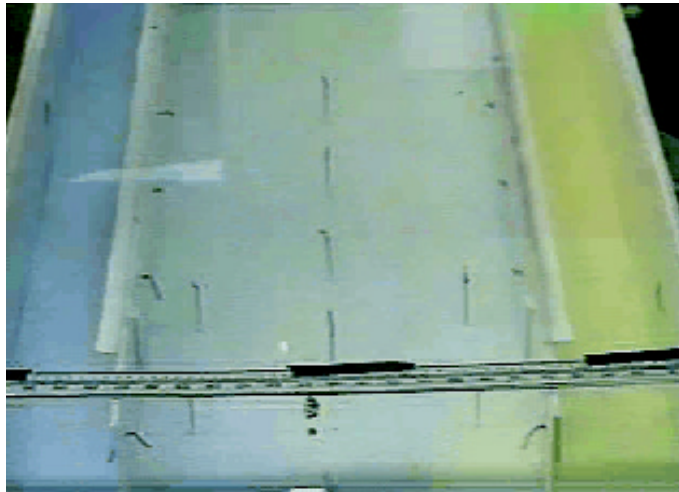


Figure 6.1 – Flow visualization in the UTATF using yarn tufts. To see video footage of the experiment, click on the image to activate the animation.

Flow Visualization in the UTATF Using Smoke Injection

In the UTATF, smoke of different colors is injected into each inlet leg of the test

section. These experiments are useful for evaluation of the mixing of the two inlet streams. The preliminary CFD models indicated well-behaved streamlines with little or no mixing between the two inlet streams beyond diffusion resulting from small turbulent fluctuations.

Flow visualization experiments in the UTATF indicate the presence of large-scale turbulent behavior in the exit channel that results in mixing of the inlet streams in a quasi-periodic manner. Indeed, the flow patterns are similar to those seen in the wake of a cylinder in cross-flow. A flow visualization experiment in the UTATF using smoke injection with a flow rate of half the nominal flow rate of 15 L/s is shown in Figure 6.2. A full speed flow visualization experiment is shown in Figure 6.3.

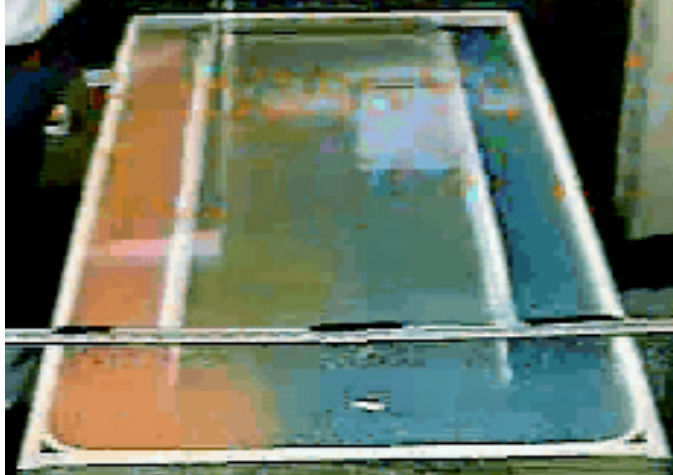


Figure 6.2 – Flow visualization experiment in the UTATF at half nominal flow. To see video footage of the experiment, click on the image to activate the animation.

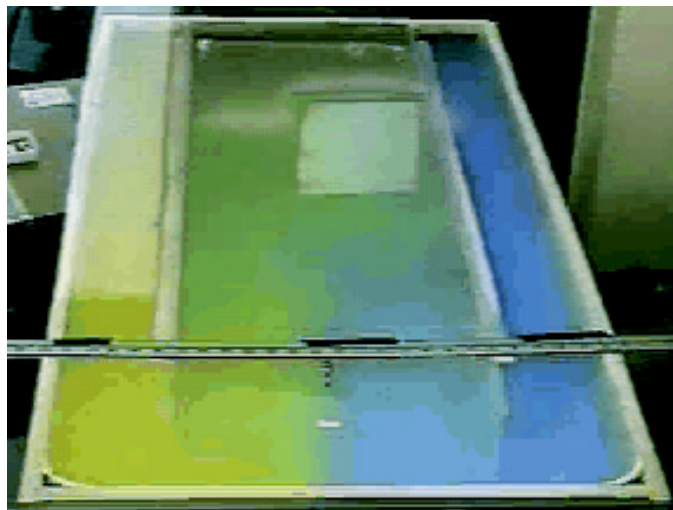


Figure 6.3 – Flow visualization experiment in the UTATF at nominal flow. To see video footage of the experiment, click on the image to activate the animation.

Flow Visualization in the UTATF Using Thermochromic Imaging

In order to better characterize the pseudo-periodic patterns seen in the smoke visualization tests, the UTATF was modified by placing thermochromic films on the bottom surface of the test section. The color of the film is indicative of the temperature of the surface of the film. By heating one of the two inlet leg flows, the mixing of the two inlet flow fields could be visualized in a more quantitative manner.

An example of thermochromic visualization experiment is shown in Figure 6.4. These experiments indicate that the flow field oscillates between three characteristic flow patterns in a pseudo-periodic manner. As seen in the CFD studies, the flow oscillates between one symmetric and two asymmetric flow patterns. The flow moves from one asymmetric pattern to the symmetric pattern to the other asymmetric pattern. The oscillation appears to always occur in this order. The period for this oscillation is between 45 and 90 seconds in the UTATF under nominal flow conditions. Figure 6.5 shows the three patterns as seen in the UTATF thermochromic imaging visualizations.

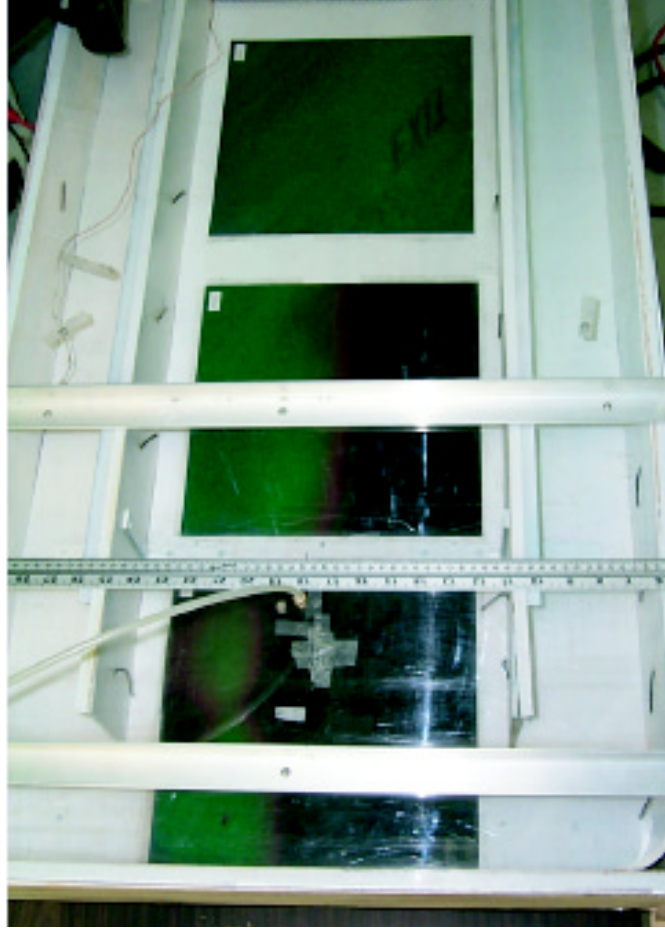


Figure 6.4 – Thermochromic imaging in the UTATF.

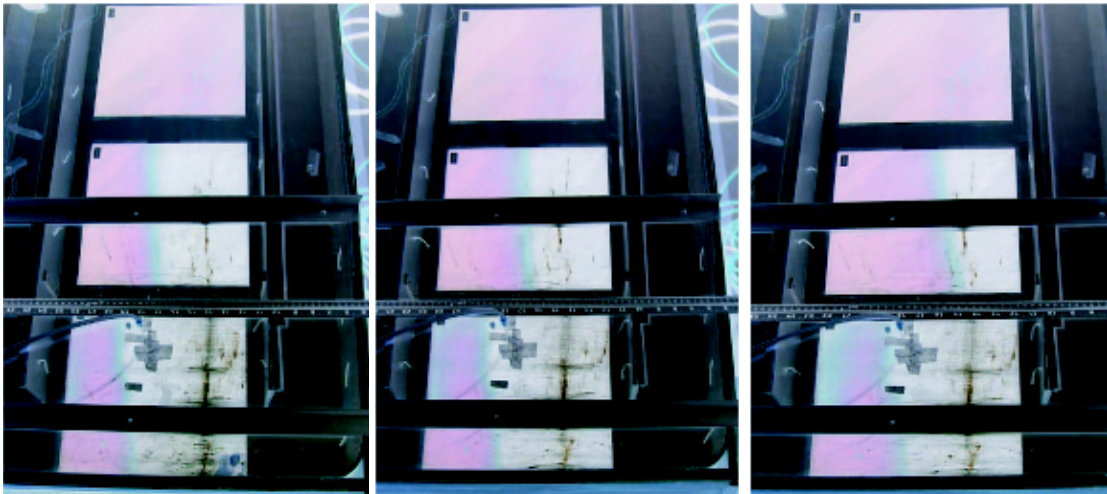


Figure 6.5 – Three characteristic modes of flow behavior in the UTATF as visualized by thermochromic imaging. The images are digitally enhanced so the boundary between the heated and unheated flow fields appears as a blue line on the thermochromic film.

6.2 Flow Visualization Studies in the WTHL Nominal Test Section

While the flow visualizations in the UTATF are good indicators of the types of phenomena that may occur in the SNS target, the UTATF is not as well scaled as the WTHL. Thus, flow visualization experiments in the WTHL provide a more reliable simulation of the flow behavior in the SNS mercury target. Two flow visualization techniques are used in the WTHL test section – bubble injection and dye injection. Bubble injection is useful for the identification of streamline and rotational behavior in the flow field and dye injection is useful for investigation of the diffusivity of the flow field.

Flow Visualization in the WTHL Using Dye Injection

Dye is injected into the test section through the openings in the test section normally used by pressure instrumentation using a nozzle that can be inserted through the wall into the flow field. A flow visualization experiment using dye injection with the injection nozzle positioned flush with the wall of the test section is shown in Figure 6.6. In this

example the flow rate through the test section is the nominal value of 11 L/s per inlet leg. The development of the stationary recirculation zones near the flow baffles, predicted by the CFD simulation, is seen in the dye visualization experiments. The third recirculation zone predicted in the target nose region of the target is not as stable as the two primary recirculation zones, and it appears and disappears at random in random positions in the nose region of the test section. Dye visualization experiments also indicate that the flow through the exit is highly

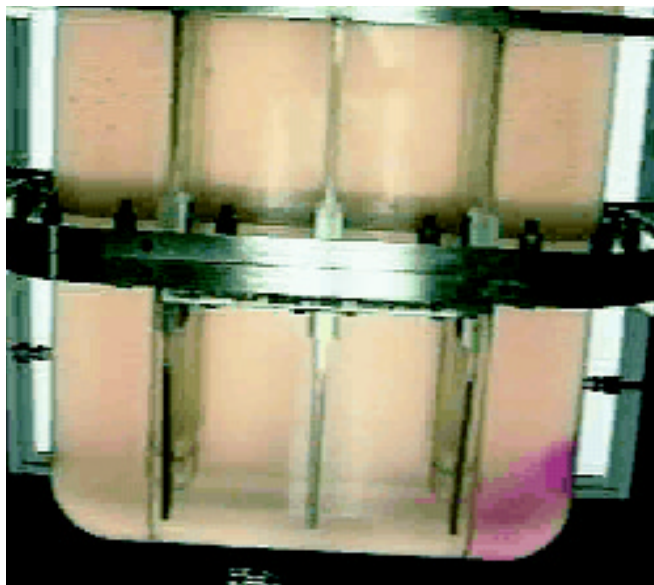


Figure 6.6 – Flow visualization in the WTHL test section at nominal flow using dye injection. To see a video recording of this experiment, click on the image to activate the animation.

diffusive downstream of the stationary recirculation zones. By the time the dye reaches the exit of the test section, it is evenly distributed across the entire width of the exit channel. Pseudo-periodic behavior like that seen in the UTATF experiments may appear in the dye visualizations in the WTHL, however the flow

is diffusive enough to render the dye relatively transparent by the time it reaches the regions where this phenomena could be clearly observed. Similar flow behavior is observed in all dye injection studies regardless of the choice of injection point. A flow visualization experiment performed at half the nominal flow rate is shown in Figure 6.7. The general flow characteristics are not drastically altered in the reduced flow rate experiments, but the flow more patterns are more easily seen in a digital image of these experiments because the residence time of the fluid in the test section is doubled.



Figure 6.7 – Flow visualization in the WTHL test section at half-nominal flow using dye injection. To see a video recording of this experiment, click on the image to activate the animation.

Flow Visualization in the WTHL Using Bubble Injection

Bubbles can be injected through any of several pressure taps in the WTHL test section using the injection nozzle from the dye injection studies. These flow visualization experiments use the nominal WTHL test section at nominal flow conditions (22 l/s). Bubble visualization clearly confirms the development of a large recirculation zone near the leading edge of the flow baffle. Additionally, bubble visualization experiments appear to indicate pseudo-periodic vortex shedding through the exit channel that originates at a point along the flow baffle just downstream from the stationary recirculation zone. This vortex shedding behavior must be assumed to play some role in the behavior seen in previous experiments in which the flow appears to alternate between three characteristic modes of flow behavior. A bubble injection visualization experiment in the nominal test section at the nominal flow rate (11 L/s per inlet leg) is shown in Figure 6.8. Similar behavior is seen in all bubble injection studies in the nominal test section regardless of the choice of injection point. A bubble injection visualization experiment in the nominal test section at half of the nominal flow rate is shown in Figure

6.9. The reduction in flow rate does not drastically alter the flow patterns, but these patterns are more easily identified in digital images at the reduced flow rate since the fluid residence time in the test section is doubled. For example, the vortex shedding mechanism is more easily observed in the reduced flow experiments.

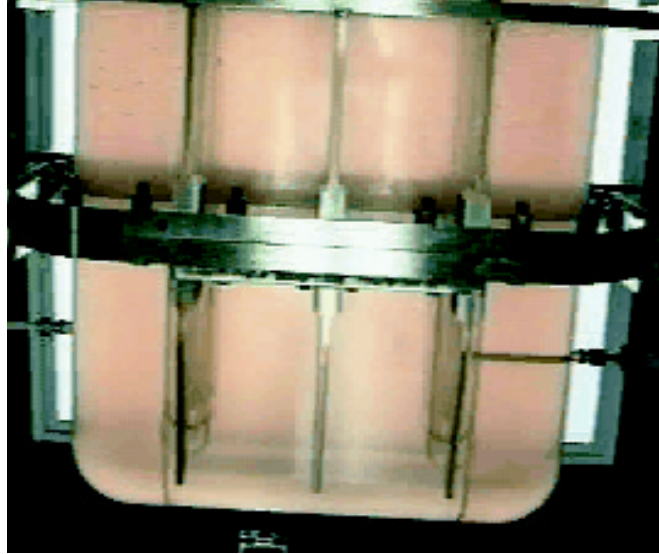


Figure 6.8 – Flow visualization in the WTHL test section at nominal flow using bubble injection. To see a video recording of this experiment, click on the image to activate the animation.

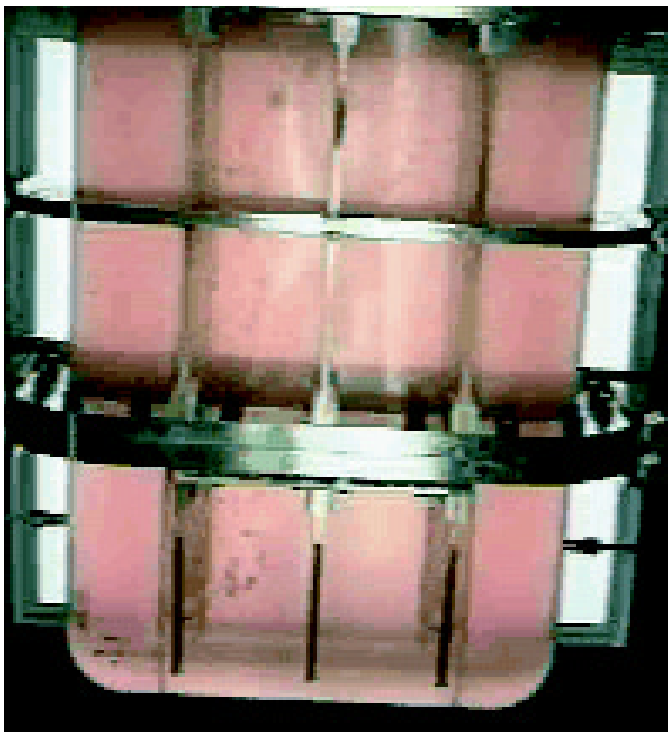


Figure 6.9 – Flow visualization in the WTHL test section at half-nominal flow using bubble injection. To see a video recording of this experiment, click on the image to activate the animation.

Chapter 7 Evaluation of Pressure Distributions and Flow Stability in the SNS Thermal Hydraulic Experiments

Flow visualization studies indicate that the flow field in the Spallation Neutron Source (SNS) target will be highly diffusive, but may exhibit large-scale turbulence structures. Furthermore, these structures may appear in a pseudo-periodic manner. If these phenomena are truly periodic, the periodic tendencies should also appear as periodic oscillations in the local pressure in the exit channel of the target. Thus, pressure studies may be used to evaluate the stability of the typical flow field in the SNS mercury target.

7.1 Evaluation of Inlet Flow Stability in the WTHL Experiments

The symmetric design of the SNS mercury target requires equal inlet flow rates in the two inlet legs that feed the bulk flow region of the target. In the Water Thermal Hydraulic Loop (WTHL) flow loop, the flow rates in the two inlet legs tend to oscillate about some mean value in a pseudo-periodic manner. This phenomenon could be completely separate from the exit channel oscillation since the simulation predicts asymmetric flow with fixed, equal inlet velocities. In order to establish an inlet flow rate that is steady (or within 1 percent of steady) in both inlets of the WTHL test section, a large pressure drop must be introduced in each inlet leg. The pressure drop is introduced by partially closing the valves used to control flow through the test section and isolate the test section from the rest of the loop during maintenance and change out activities. Since the current design of the SNS target system does not include control valves on the two inlet legs in an effort to reduce maintenance requirements, the pressure drop needed to stabilize the flow must be well characterized.

As shown in Figure 7.1, three pressure gauges, a manometer open to the atmosphere, and a manometer that measures the differential pressure in the test section were used to characterize the pressure losses in the WTHL test facility. The local pressures

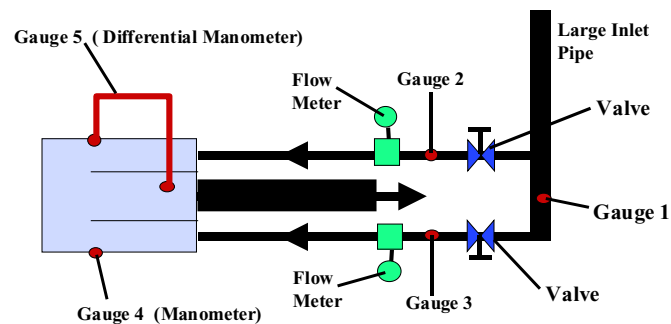


Figure 7.1 – Pressure instrumentation used in the evaluation of inlet flow stability in the WTHL.

in the WTHL test section do not match those that are expected in the actual target. Therefore, a scaling relationship that allows the WTHL data to be used to predict the pressure drop needed in the actual system must be found. The ratio of the inlet pressure drop needed to establish steady flow (variation <1 percent) to the pressure drop in the test section was plotted versus the Reynolds number and the dynamic head in an effort to establish such a relationship. The pressure drop ratio is shown versus Reynolds number in Figure 7.2 and versus dynamic head in Figure 7.3.

Additional data is needed to extrapolate the results of these studies to the much higher Reynolds numbers found in the SNS mercury target and to evaluate the effectiveness of the pressure drop ratio as a scaling parameter. Since the WTHL facility is not well scaled for experiments that require a high degree of similarity in the Reynolds number of the test section and the Reynolds number of the SNS mercury target, pressure drop studies should be included in test plans for future mercury experiments. However, initial data from the WTHL experiments appears to indicate that the value of pressure drop ratio must be on the order of 20 for stable steady flow to be established in the two inlet legs.

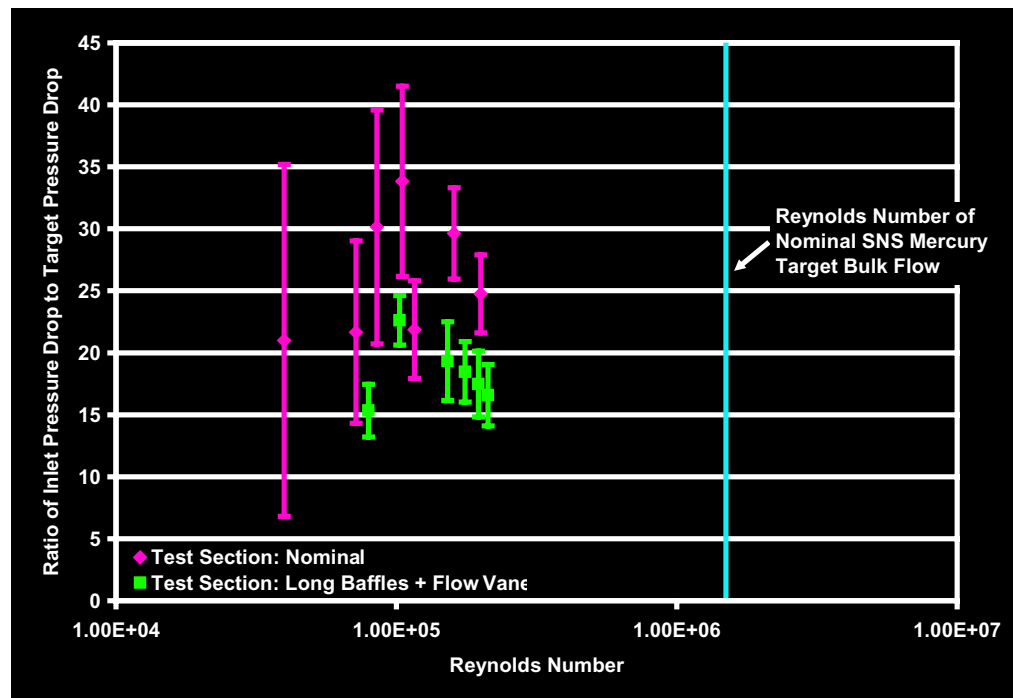


Figure 7.2 – Pressure drop ratio versus Reynolds number.

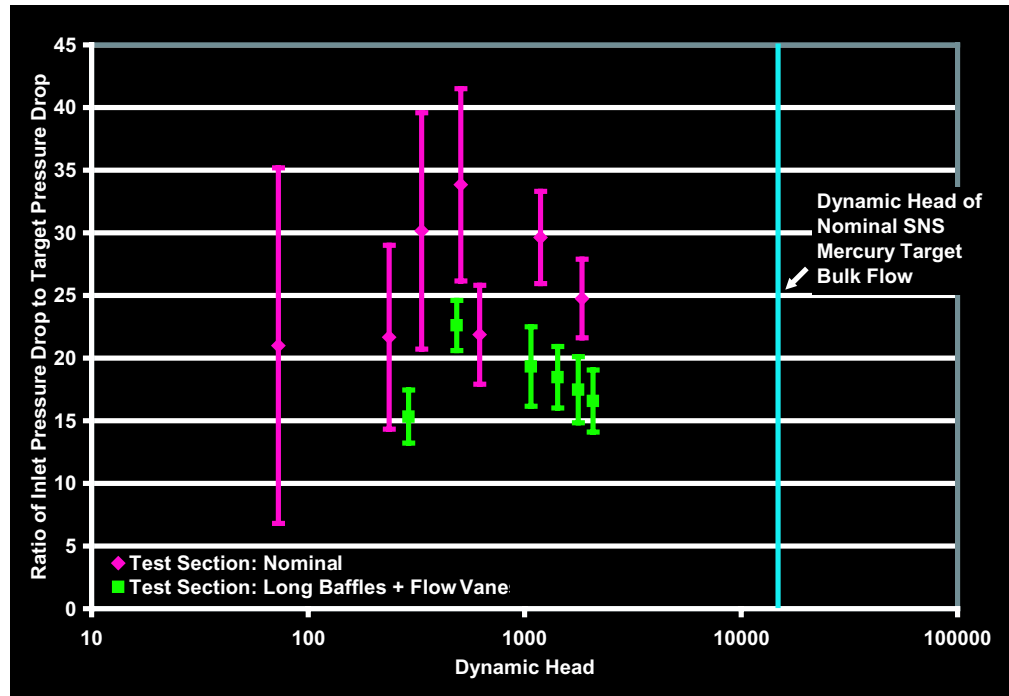


Figure 7.3 – Pressure drop ratio versus the dynamic head.

7.2 Evaluation of Periodicity of Pressure Oscillations in the UTATF

In the flow visualization experiments in the University of Tennessee Air Test Facility (UTATF), the flow field was observed to oscillate between three characteristic modes of flow behavior. The mechanism that drives the oscillation appears to operate in a pseudo-periodic manner. Such periodicity should be evident in the pressure oscillations in the exit channel of the experimental test section.

Using the single instrument port, the local pressure was monitored at two positions along the centerline of the exit channel in the UTATF. The first position is located at the centerline of the exit channel and 1 inch in front of the leading edges of the flow baffles. The second is located at a position on the centerline of the exit channel that is 3 inches downstream of the leading edges of the flow baffles. The time dependent pressure oscillations at these locations are shown in Figure 7.4 and Figure 7.5.

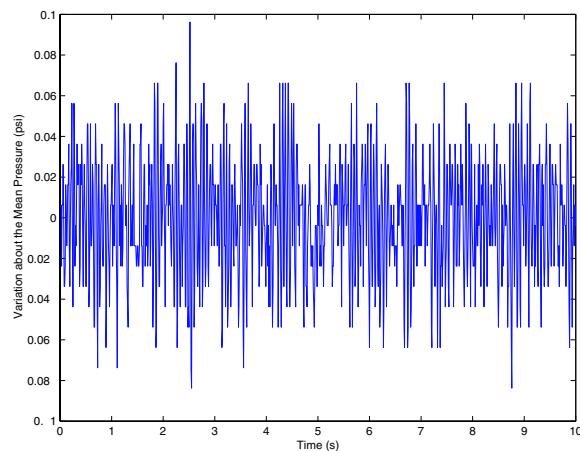


Figure 7.4 Variations in local pressure in the UTATF at a position located on the centerline of the exit channel that is 1 inch upstream of the leading edge of the flow baffles.

Evaluation of periodicity in such a flow is difficult because the inherent randomness of a turbulent flow field introduces some randomness into otherwise periodic phenomena. In order to evaluate the periodicity of the pressure oscillations in the UTATF, the short-term Fourier transform is calculated for each of the pressure signals shown. The short-term Fourier transform is calculated by dividing the signal into several overlapping windows and then performing a fast Fourier transform on the data contained in each window. The results of these evaluations are shown in graphical form in Figure 7.6 and Figure 7.7. In these representations, dark red indicates

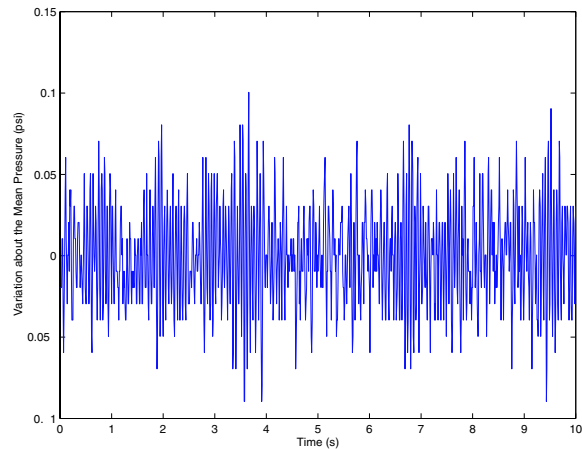


Figure 7.5 Variations in local pressure in the UTATF at a position located on the centerline of the exit channel that is 3 inches downstream of the leading edge of the flow baffles.

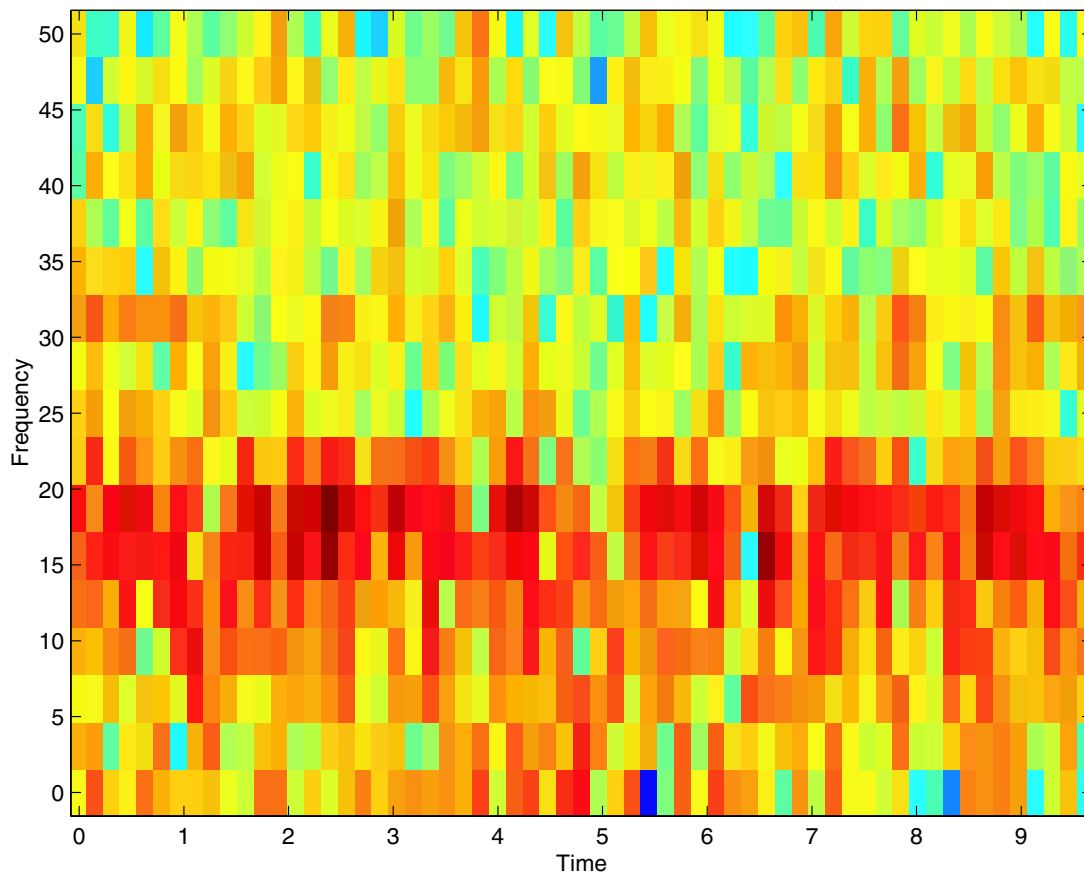


Figure 7.6 – Short-term Fourier transform of the pressure signal shown in Figure 7.4.

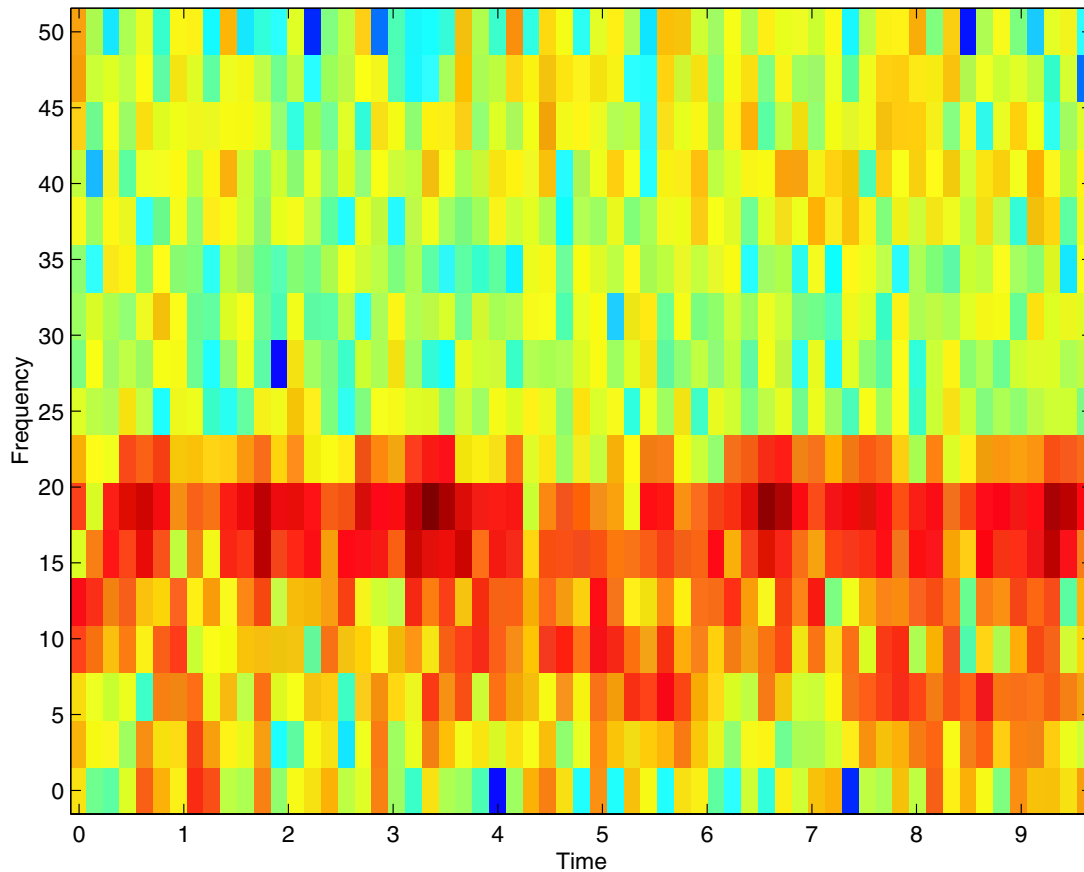


Figure 7.7 – Short-term Fourier transform of the pressure signal shown in Figure 7.5.

maximum frequency gains and blue indicates minimums. The most common periodic phenomena appear to occur at a frequency of 20 Hz, which corresponds to the frequency of the blower used to pull the air through the system. Interestingly, both measurements show some indications of lower frequency phenomena. These lower frequencies could possibly result from resonance behavior within the test section, or they could be indicative of the lower frequency flow behaviors observed in the flow visualization experiments.

7.3 Evaluation of the Pressure Oscillations in the WTHL

As demonstrated by the previous studies, the pressure drop in the mercury target, as predicted by the CFD simulations, is a critical design criterion for the target and its associated systems. Therefore, the ability of the simulation to predict the pressure drop should be verified just as the ability of the simulation to predict velocity information should be verified. To provide more rigorous data for this verification, the pressure drops in the nominal test section was evaluated using five

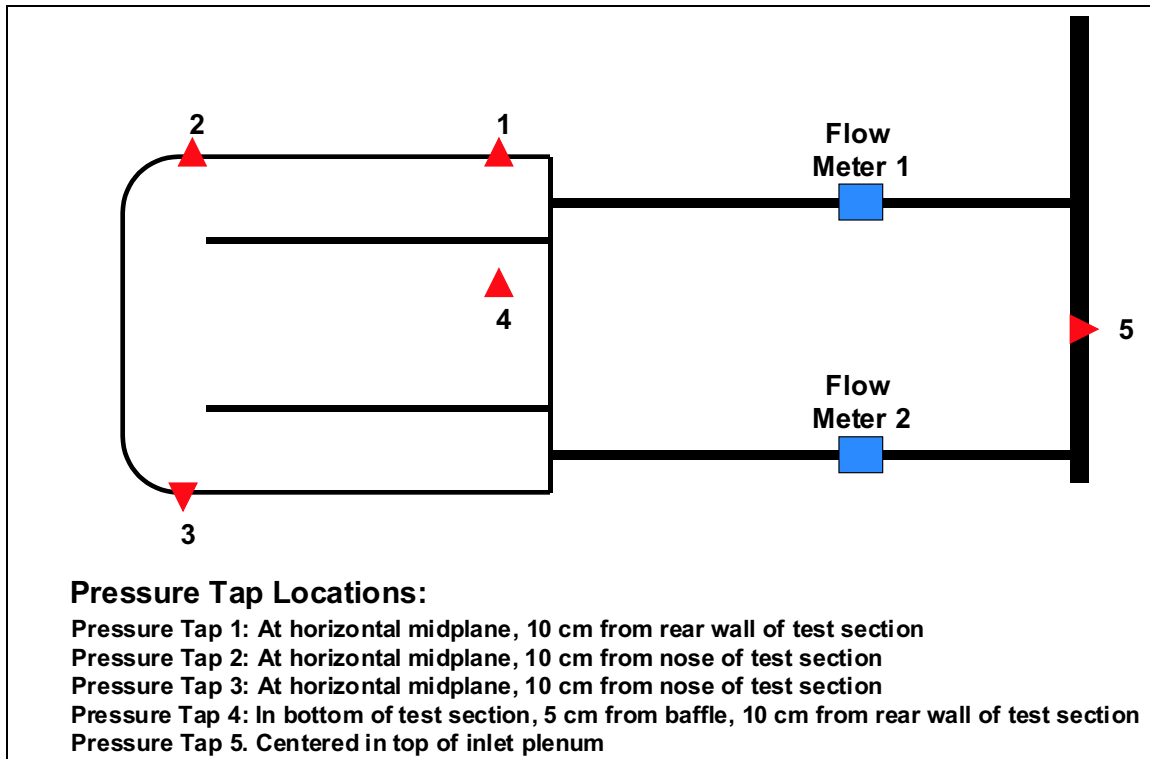


Figure 7.8 – Pressure tap locations for evaluation of test section pressure losses.

pressure transducers, located as shown in Figure 7.8. The pressure drop in the test section was evaluated for three different flow rates, 11, 7.14, and 12.75 L/s per inlet leg. Results are shown in Table 7.1 for each pressure measurement and the calculated pressure drop. The reported error is the observed variation in the measurement resulting from turbulent flow behavior. The variation in the flow rate in these experiments is approximately 1 percent.

Table 7.1 – Pressure drop data for nominal test section. Time-dependent variations are reported as percent for flow rate measurements and psig for pressure measurements.

Flow Rate (l/s) Per Inlet Leg	PT-1 (psig)	PT-2 (psig)	PT-3 (psig)	PT-4 (psig)	PT-5 (psig)	Average Test Section Pressure Drop (psi)
11.0 +/- 1%	1.24 +/- 0.15	1.24 +/- 0.20	1.19 +/- 0.20	0.87 +/- 0.20	13.0 +/- 0.75	0.37
7.14 +/- 1%	0.64 +/- 0.15	0.59 +/- 0.10	0.58 +/- 0.10	0.47 +/- 0.10	10.0 +/- 0.50	0.17
12.75 +/- 1%	1.64 +/- 0.25	1.64 +/- 0.25	1.59 +/- 0.20	1.27 +/- 0.20	20.0 +/- 2.0	0.37

Chapter 8 Evaluation of Time-Averaged Characteristics of the Velocity Field in the SNS Thermal Hydraulic Experiments

The ability of the Computational Fluid Dynamics (CFD) simulation to accurately predict temperature distributions in the SNS mercury target is completely dependent on the ability of the fluid dynamics models to predict the velocity field in the target. The steady state simulations on which the temperature distribution calculations are based are essentially time-averaged portraits of the turbulent flow behavior in the mercury target. Thus, the velocity field in each of the thermal hydraulic experiments is mapped to provide time-averaged data that can easily be used to evaluate the ability of the CFD model to predict the velocity field in those experiments.

8.1 Mapping the Time-Averaged Velocity Field in the UTATF

The velocity profiles in the University of Tennessee Air Test Facility (UTATF) are mapped by positioning a single hot wire anemometer in the instrumentation port of the test section. The instrumentation port is repositioned for each measurement so that the magnitude of the total flow velocity is recorded on a 1-inch grid in the nose region of the test section. Additionally, the velocity in the UTATF test section is mapped at three different depths in the test section – 1.5 inches above the centerline, at the centerline and 1.5 inches below the centerline. In these experiments, the time-averaged velocity and the standard deviation of the velocity were recorded. Figures 8.1, 8.2, and 8.3 show maps of the velocity profiles measured at the three specified depths in the UTATF test section. Since the UTATF is not well scaled to the SNS mercury target, this data is not ideal for qualification of the CFD code. However, it does appear to confirm the time averaged flow behavior predicted by the CFD code.

8.2 Mapping the Time-Averaged Velocity Field in the WTHL Using Laser Doppler Velocimetry

The mapping of the velocity field in the WTHL test section provides quantitative data for validation of the CFD simulations used in the development of the SNS mercury target. The mapping of the velocity field is accomplished using a

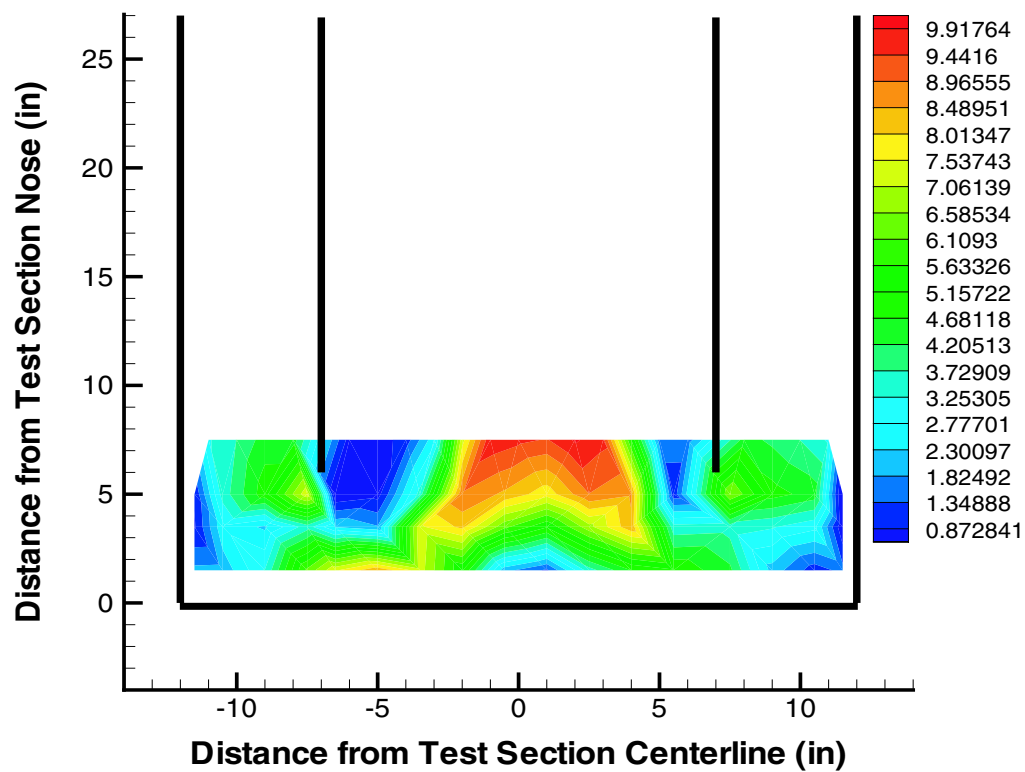


Figure 8.1 – Time-averaged total velocity magnitude in the UTATF test section at 1.5 inches above the centerline.

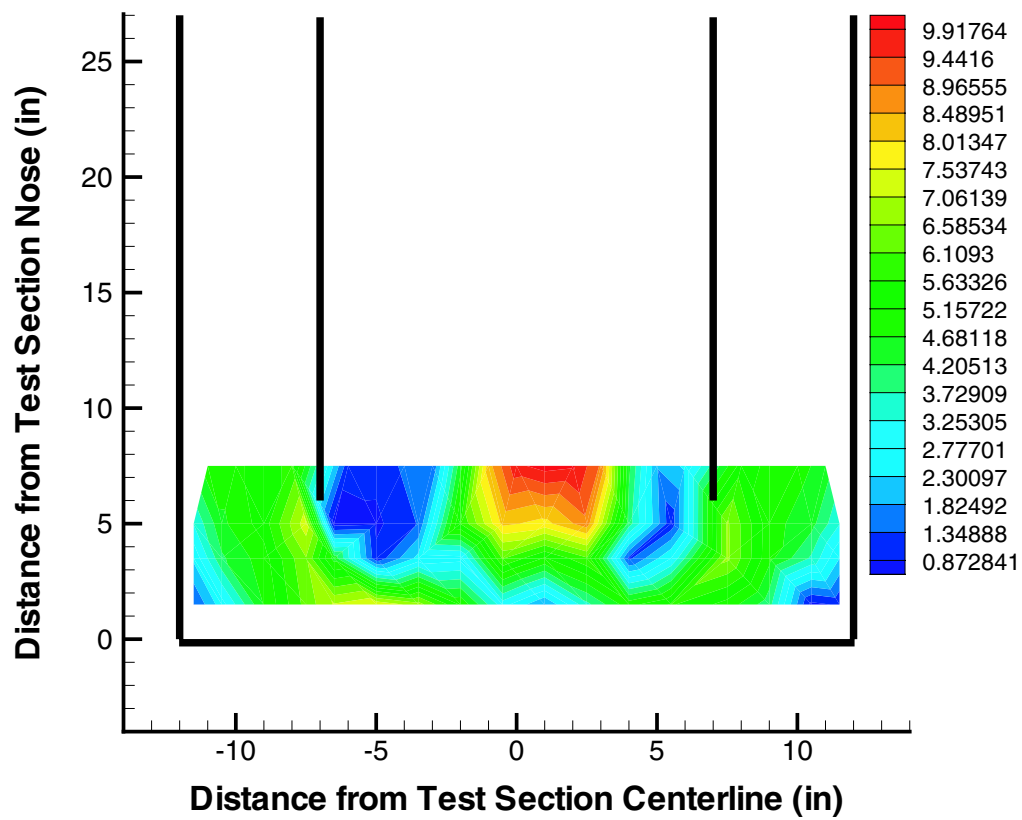


Figure 8.2 – Time-averaged total velocity magnitude in the UTATF test section at the centerline.

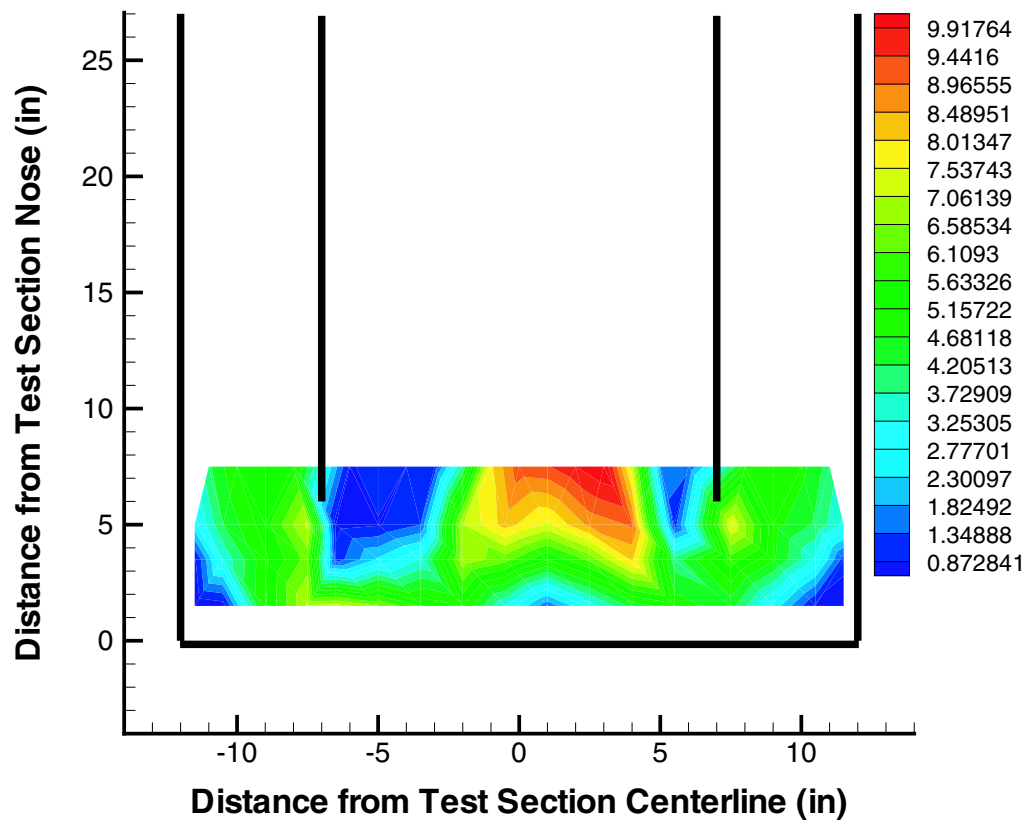


Figure 8.3 – Time-averaged total velocity magnitude in the UTATF test section at 1.5 inches below the centerline.

two-dimensional Laser Doppler Velocimeter (LDV) to measure the two horizontal (x and y) components of the flow velocity at various points throughout the WTHL test section. As many as four thousand individual measurements are made at each location over a period of 1 to 10 seconds and the system software reports the time-averaged values calculated from those individual measurements. The velocity mapping is confined to the flow regions visible through the two Plexiglas windows machined into the top surface of the WTHL test section. The windows are used because the optical quality of the polyurethane from which the majority of the test section is manufactured is not sufficient to allow quality LDV measurements. Preliminary LDV measurements used a random sampling technique to decrease the time required to obtain a data set. When this technique is used, the component-wise measurements are made at random, not simultaneously. While this technique does not affect the accuracy of the measurement of each component of the flow, the calculated total velocities and total turbulence intensities are based on the average value of a random sample of each component. The error associated with these total values can be reduced if a coincidence sampling method is used so that the component-wise measurements are made simultaneously. Furthermore, the coincidence sampling method may allow the LDV data to be used to evaluate time dependent flow behaviors. Therefore, the coincidence sampling method is used in the collection of all LDV data presented herein.

LDV Velocity Measurements in the WTHL Test Section

LDV mapping has been completed in the region of the test section visible through the acrylic window nearest the test section nose in both the nominal test section and the alternate test section at the nominal flow velocity (11 l/s per inlet leg). Additional LDV mapping has been completed in the nominal test section in the region visible through the second acrylic window. Data is collected at five different depths in the regions visible through the two acrylic windows:

1. 10 mm from the upper inside surface of the test section
2. 20 mm from the upper inside surface of the test section
3. the vertical centerline of the target flow field
4. 20 mm from the bottom inside surface of the test section
5. 10 mm from the bottom inside surface of the test section.

Figures 8.4 through 8.13 show the total velocity vectors and the magnitude of the turbulence intensity measured in the front window of the nominal test section for each of the prescribed depths. Velocity and turbulence intensity data collected in

both windows of the nominal test section are presented in Figures 8.14 through 8.23. Additional velocity maps from these experiments may be found in Appendix A. The laser doppler velocimetry data consistently indicates that the flow field is relatively symmetric on a time averaged basis. The measured velocity profiles indicate the development of two primary recirculation zones as the flow field separates from the surface at the leading edge of the flow baffle. A higher velocity jet is also seen between these two regions, as expected. As a result of the separation from the wall, the flow is much more turbulent in the recirculation regions than in the rest of the flow field, regardless of depth in the test section.

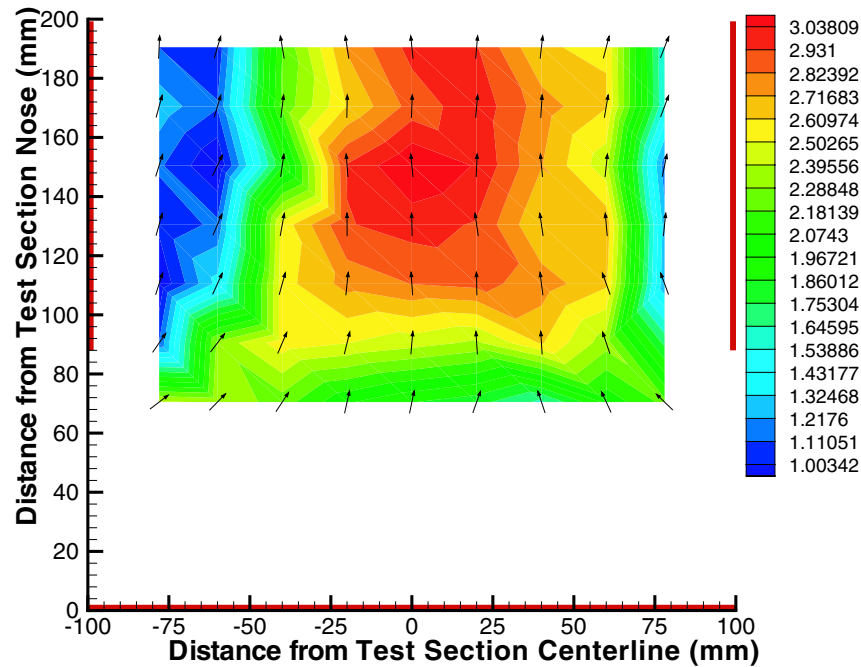


Figure 8.4 – Velocity in front window of the nominal test section at a depth of 10 mm from the upper inside surface.

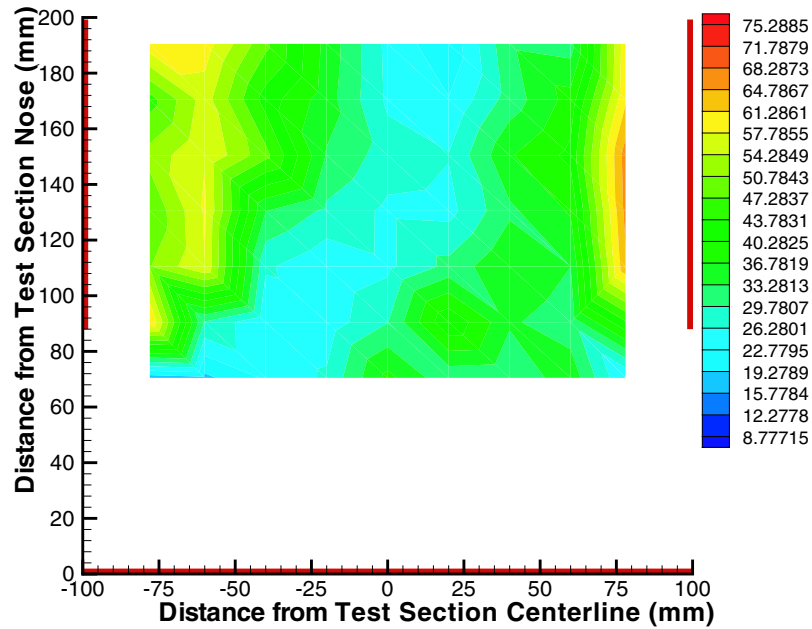


Figure 8.5 – Turbulence intensity in the front window of the nominal test section at a depth of 10 mm from the upper inside surface.

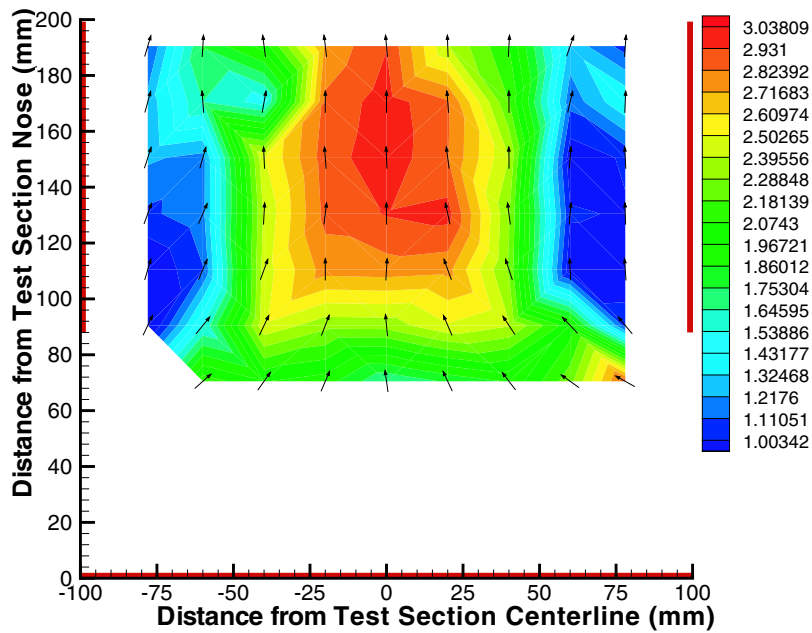


Figure 8.6 – Velocity in front window of the nominal test section at a depth of 20 mm from the upper inside surface.

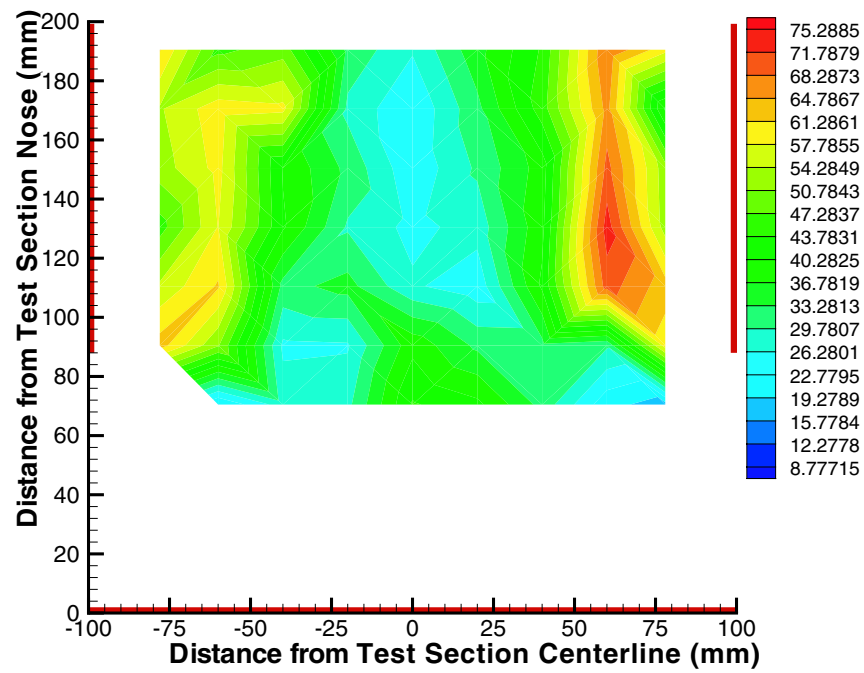


Figure 8.7 – Turbulence intensity in front window of the nominal test section at a depth of 20 mm from the upper inside surface.

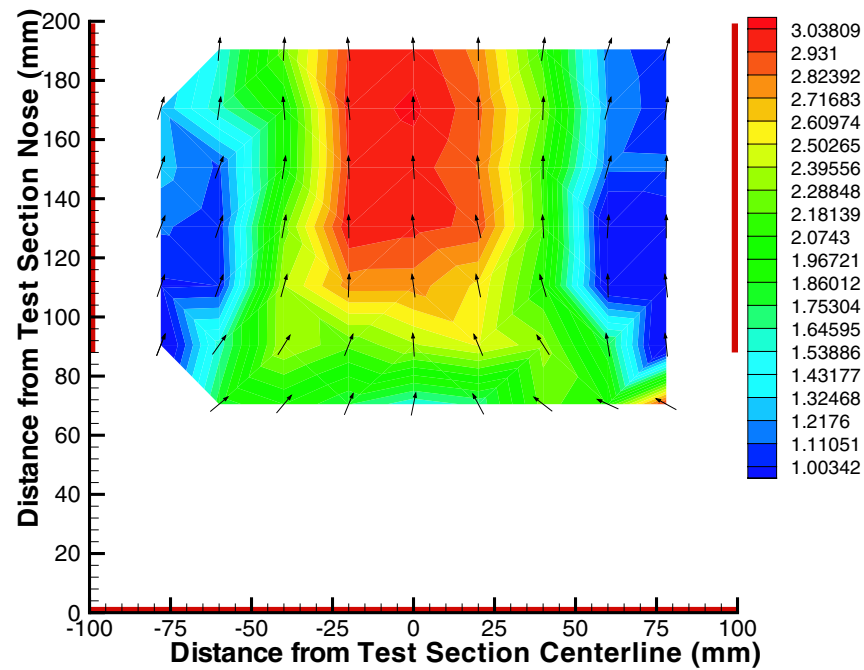


Figure 8.8 – Velocity in front window of the nominal test section at the horizontal midplane.

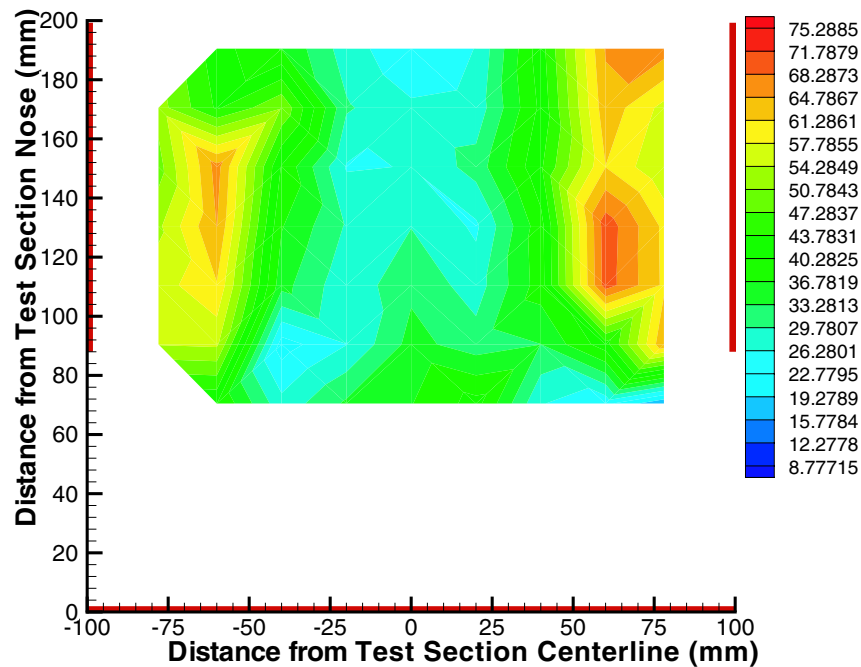


Figure 8.9 – Turbulence intensity in front window of the nominal test section at the horizontal midplane.

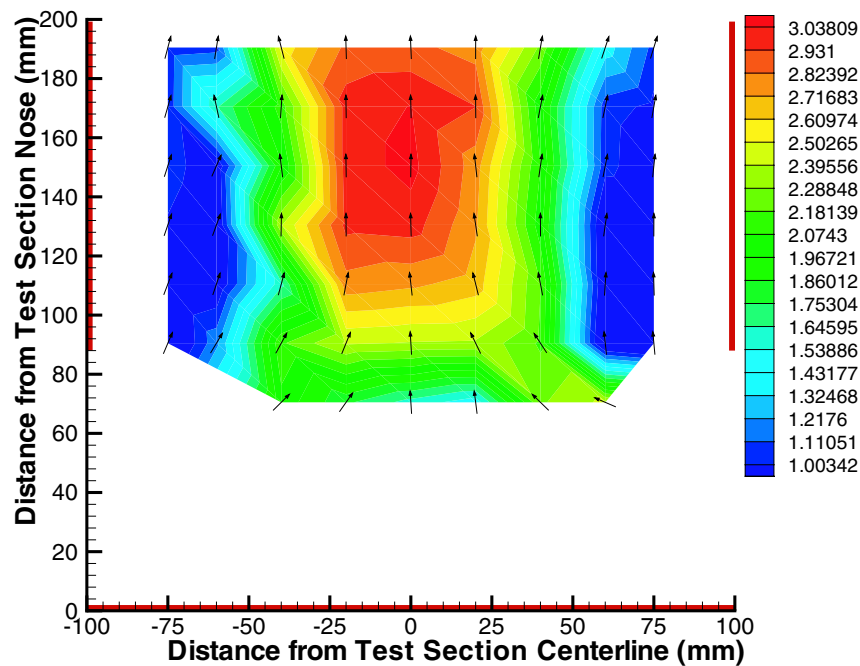


Figure 8.10 – Velocity in front window of the nominal test section at a depth of 20 mm from the lower inside surface.

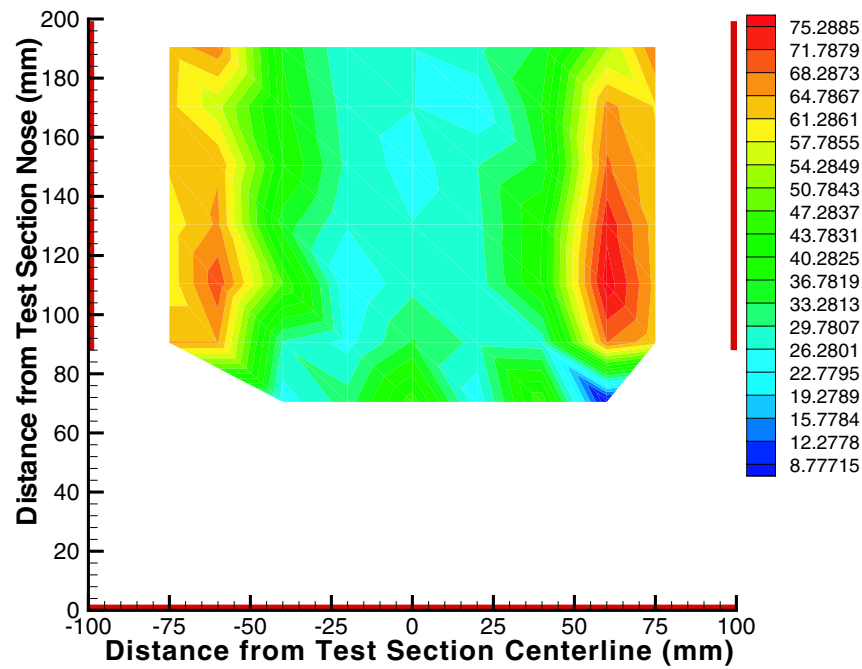


Figure 8.11 – Turbulence intensity in front window of the nominal test section at a depth of 20 mm from the lower inside surface.

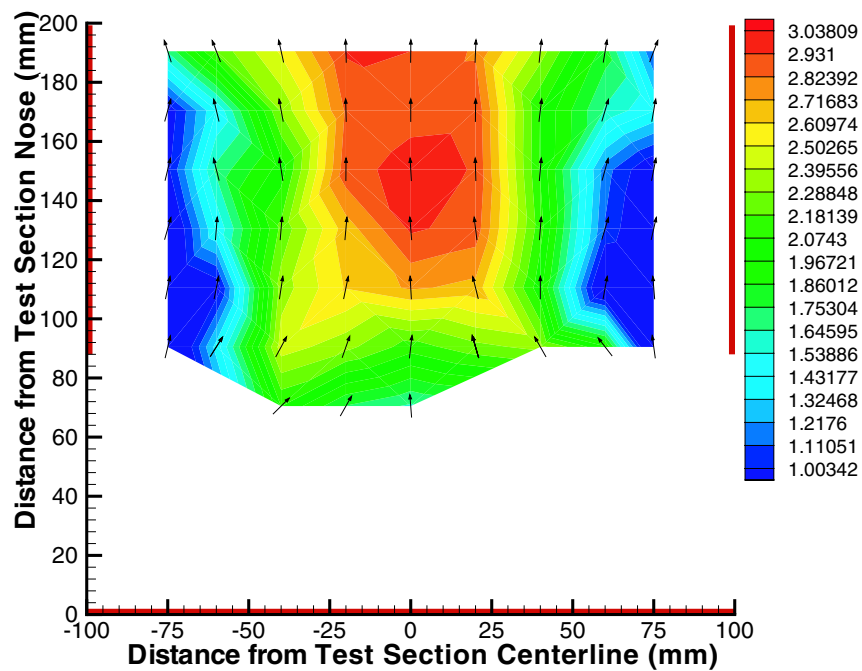


Figure 8.12 – Velocity in front window of the nominal test section at a depth of 10 mm from the lower inside surface.

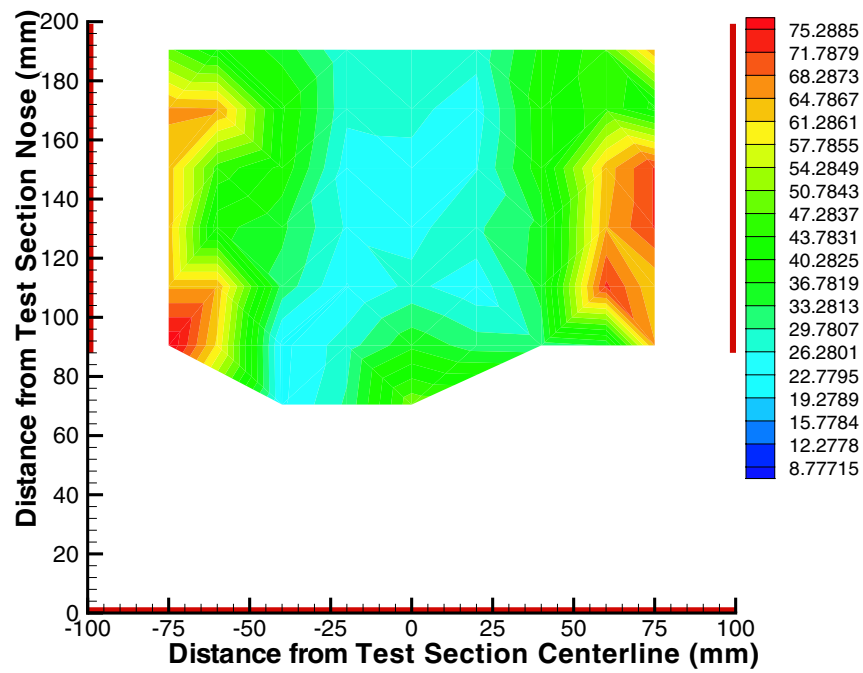


Figure 8.13 – Turbulence intensity in front window of the nominal test section at a depth of 10 mm from the lower inside surface.

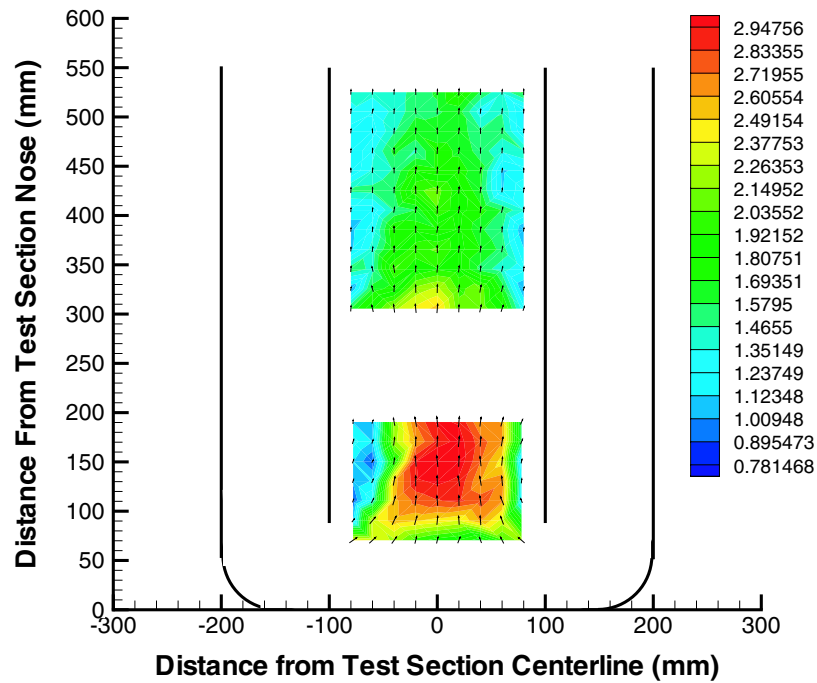


Figure 8.14 – Velocity in nominal test section at a depth of 10 mm from the upper inside surface.

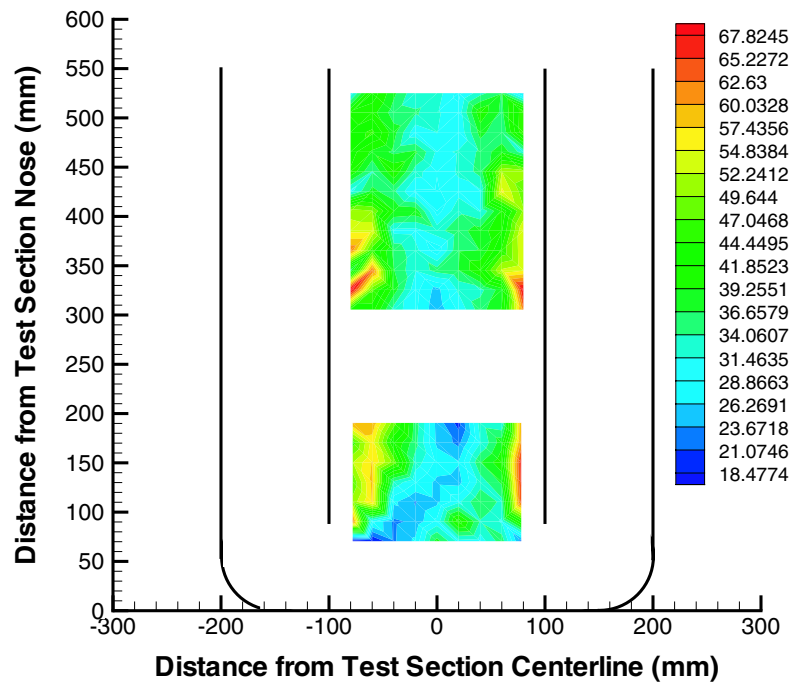


Figure 8.15 – Turbulence intensity in nominal test section at a depth of 10 mm from the upper inside surface.

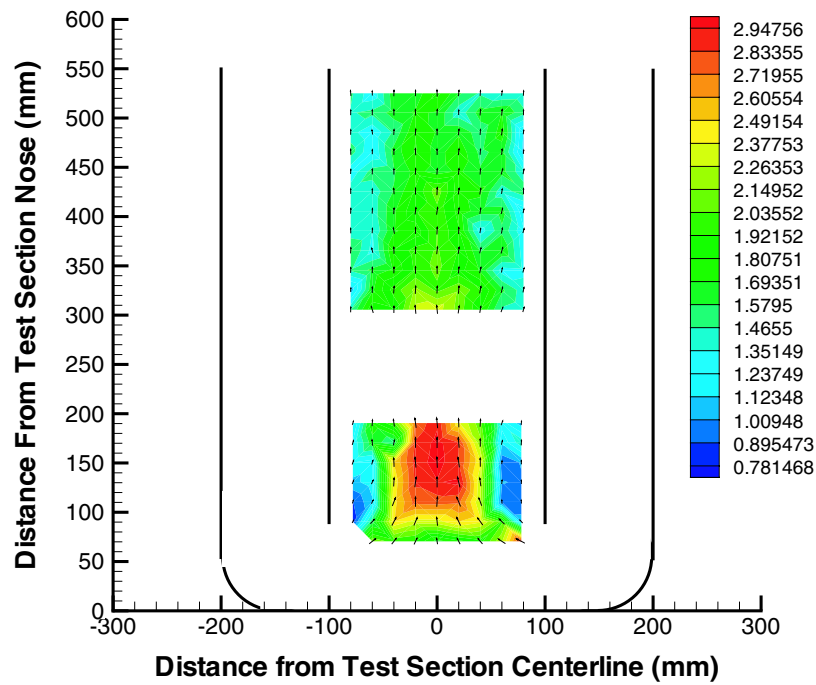


Figure 8.16 – Velocity in nominal test section at a depth of 20 mm from the upper inside surface.

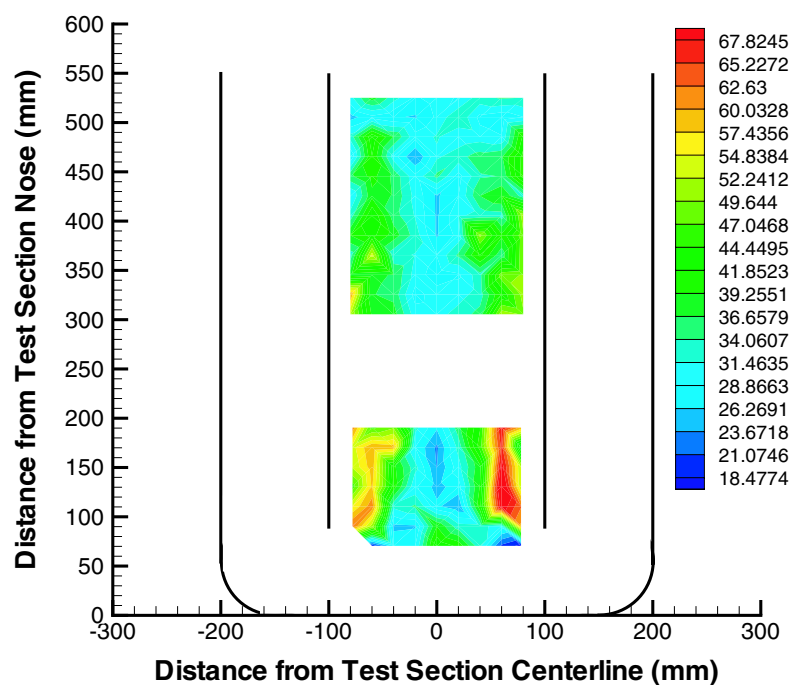


Figure 8.17 – Turbulence intensity in nominal test section at a depth of 20 mm from upper inside surface.

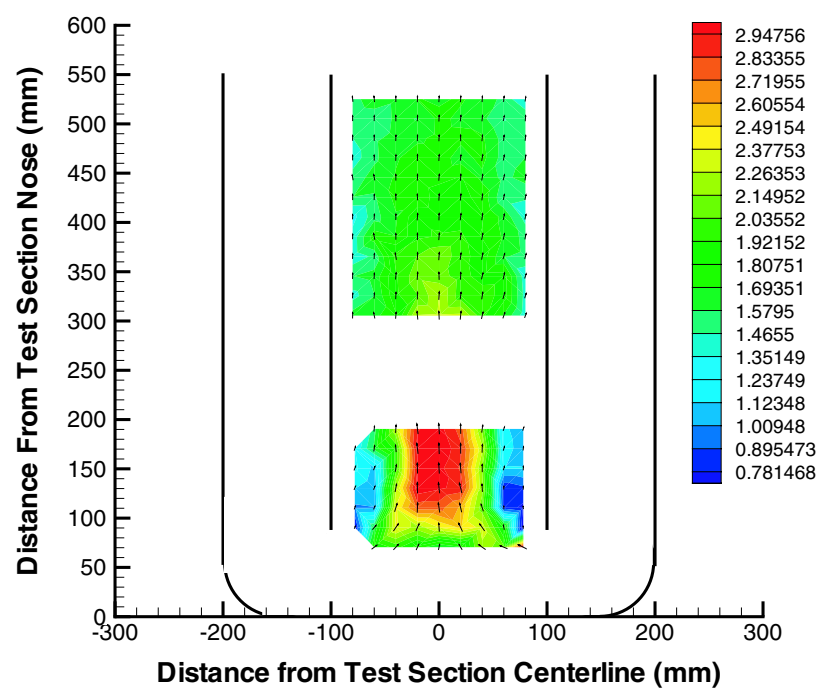


Figure 8.18 – Velocity in nominal test section at horizontal midplane.

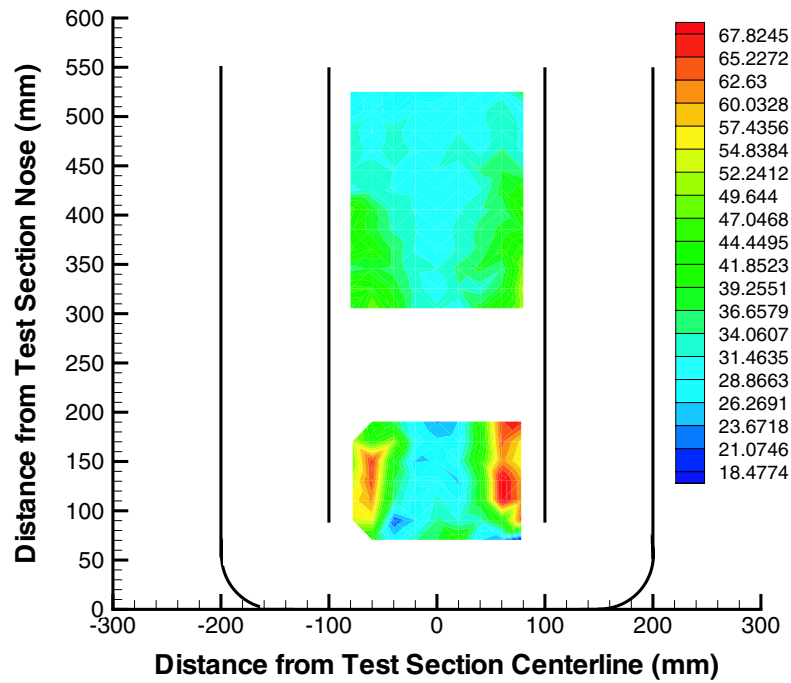


Figure 8.19 – Turbulence intensity in nominal test section at horizontal midplane.

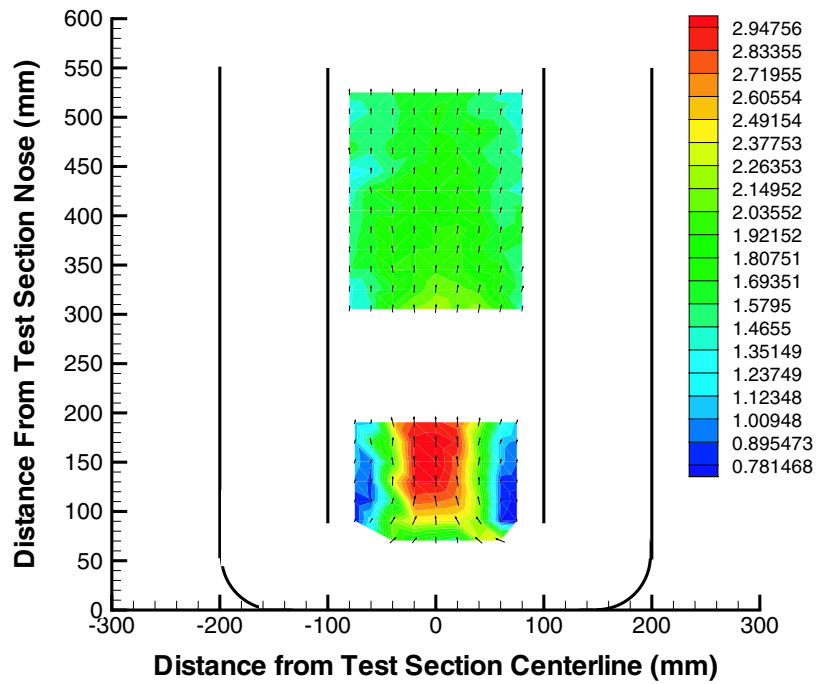


Figure 8.20 – Velocity in nominal test section at a depth of 20 mm from the lower inside surface.

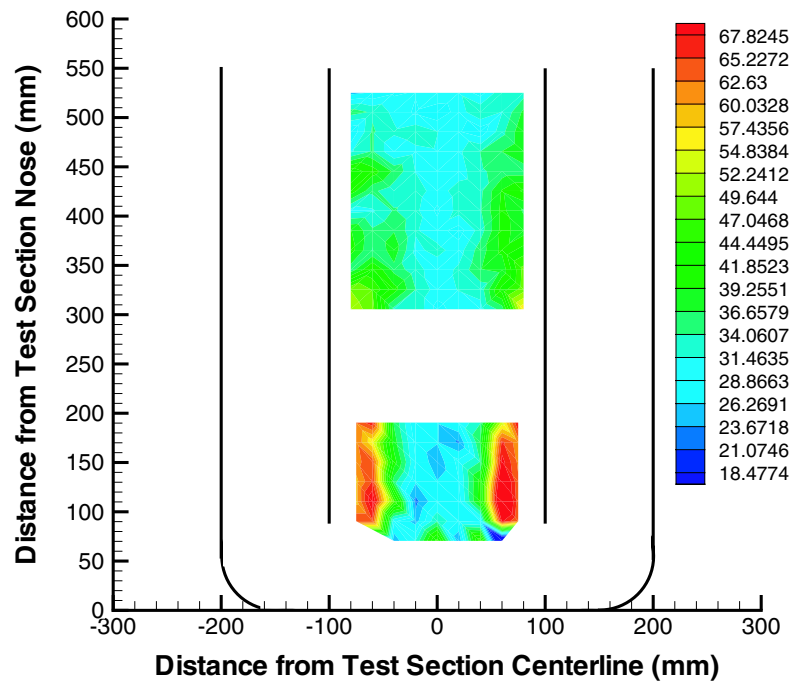


Figure 8.21 – Turbulence intensity in nominal test section at a depth of 20 mm from the lower inside surface.

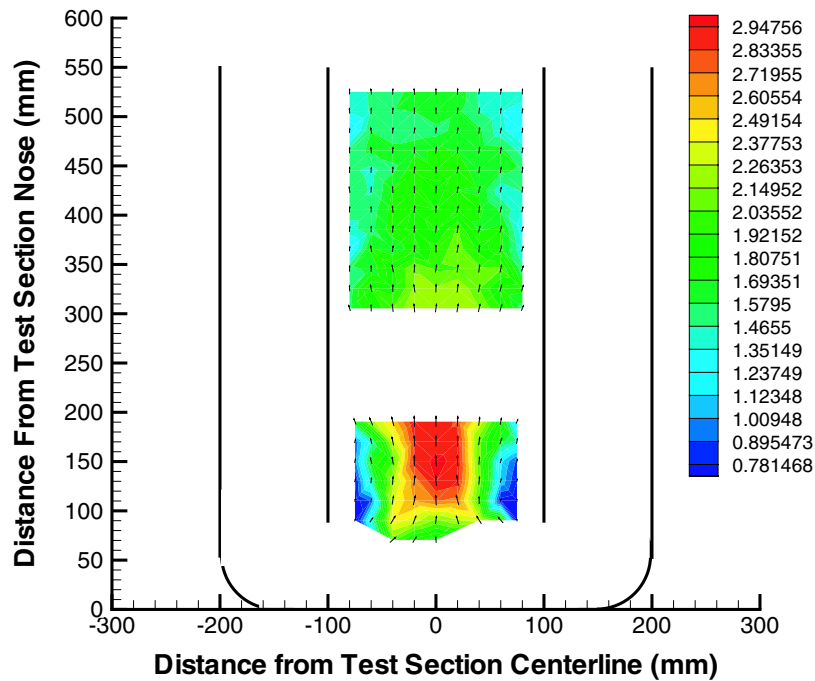


Figure 8.22 – Velocity in nominal test section at a depth of 10 mm from lower inside surface.

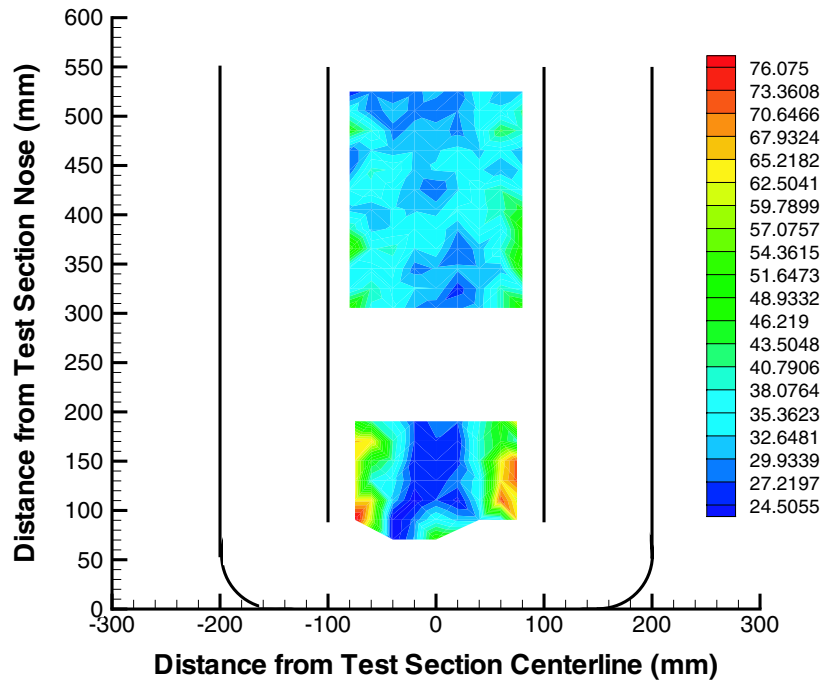


Figure 8.23 – Turbulence intensity in nominal test section at a depth of 10 mm from lower inside surface.

Repeatability of LDV Measurements in the WTHL Test Section

Multiple data sets were recorded at the horizontal mid-plane of the test section in order to evaluate the repeatability of the experiment. The measured velocities are plotted together for comparison in Figures 8.24 through 8.27 for the nominal test section. Each figure indicates the measured total velocity as a function of distance from the vertical mid-plane at one of four distances (70.5, 110.5, 150.5, and 190.5 mm) from the test section nose. The error bars indicate the standard deviation in the velocity measurement at each measurement position. The standard deviation is indicative only of the fluctuation during the period of measurement at each point, not the error associated with each measurement. The error associated with the measurement itself is negligible in comparison to these fluctuations. While the error associated with the motion of the traverse system is negligible, the spatial error resulting from the orientation of the test section and the traversing system is approximately ± 2 mm. As seen in the graphical comparisons, the differences between most of the data points are well within the bounds of the variations about any given data point.

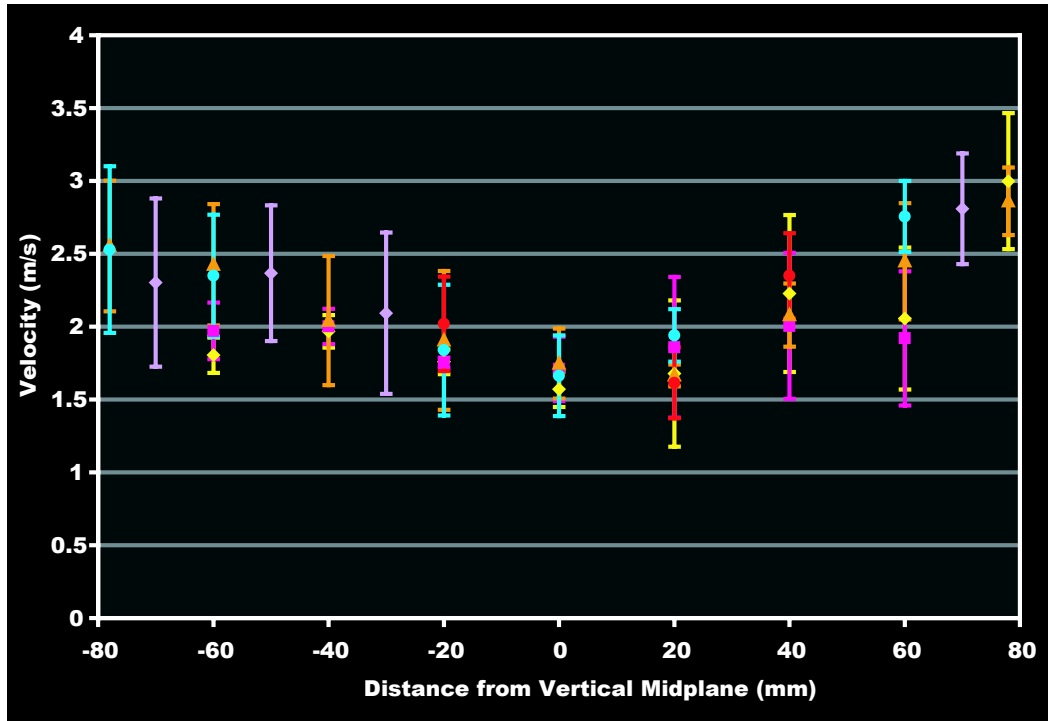


Figure 8.24 – Comparison of velocity measurements at horizontal midplane and a distance of 70.5 mm from the test section nose in nominal test section.

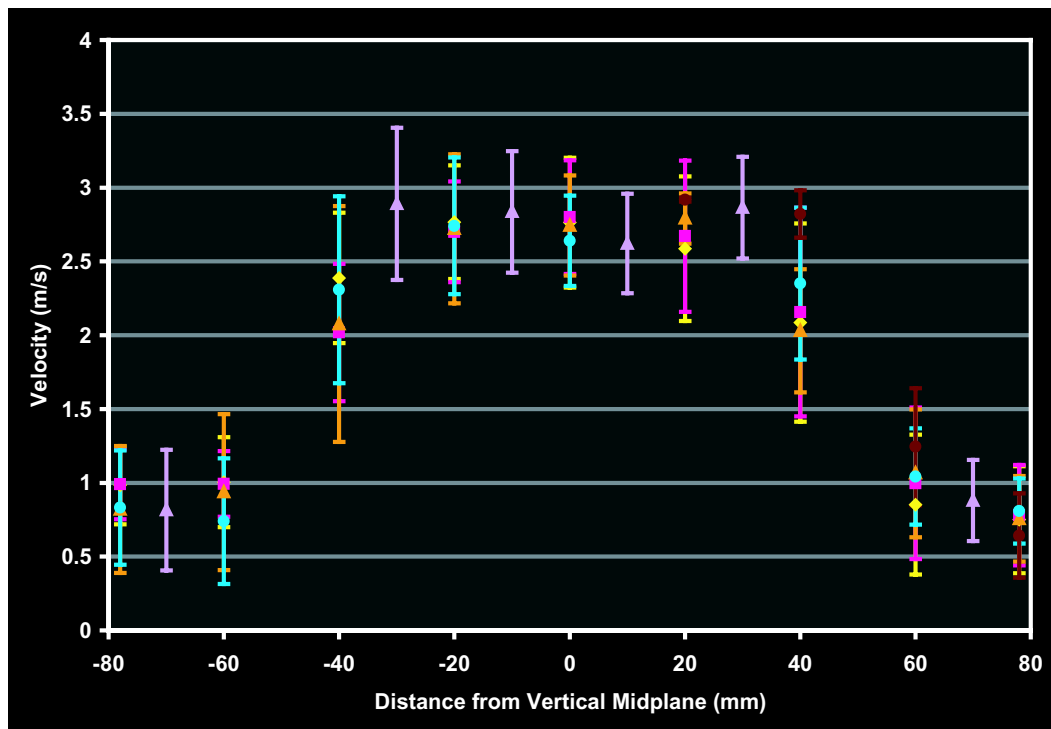


Figure 8.25 – Comparison of velocity measurements at horizontal midplane and a distance of 110.5 mm from the test section nose in nominal test section.

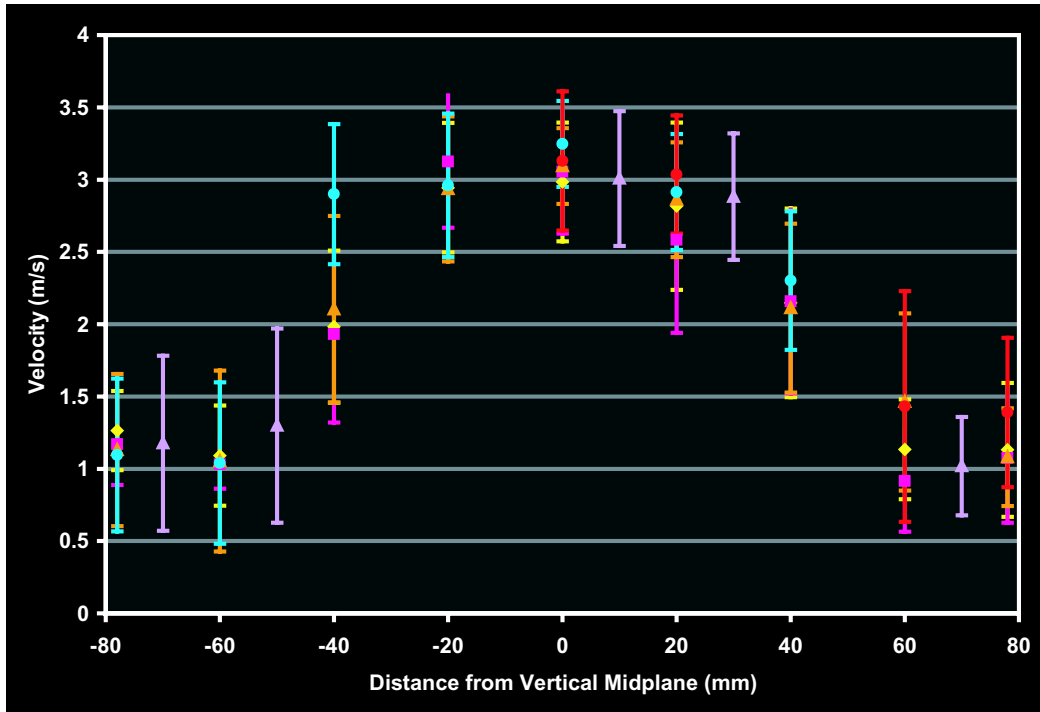


Figure 8.26 – Comparison of velocity measurements at horizontal midplane and a distance of 150.5 mm from the test section nose in nominal test section.

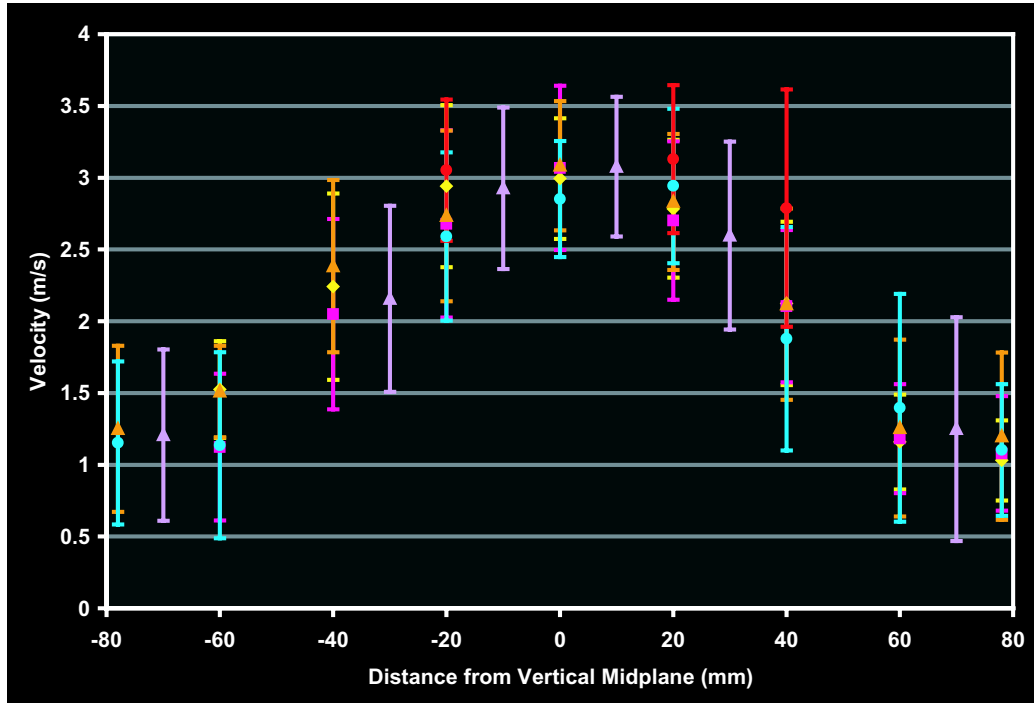


Figure 8.27 – Comparison of velocity measurements at horizontal midplane and a distance of 190.5 mm from the test section nose in nominal test section.

In order to provide a more thorough verification of the repeatability of the LDV measurements, additional velocity data was collected at the horizontal midplane of the nominal WTHL test section at two additional flow rates. First, two data sets were collected at half the nominal flow rate, 5.5 L/s per inlet leg. The comparison of these data sets are shown in Figures 8.28 through 8.30. Second, two data sets were collected at 70 percent of the nominal flow rate, 7.7 L/s per inlet leg. The comparison of the measurements from these data sets are shown in Figures 8.31 through 8.34. The velocity maps from both data sets can be found in Appendix A.

8.3 Additional Velocity Measurements in the WTHL Nominal Test Section Using Ultrasonic Velocity Profiling

The Ultrasonic Doppler Profiler (UDP) is a second instrument available in the WTHL experiments for the evaluation of the velocity field. In contrast to the LDV which makes measurements at a single point, the UDP has the ability to measure the velocity at several positions along a line simultaneously. The UDP is also not as limited by the material properties of the WTHL test section, and it can be used to make measurements very near the surface of the flow baffles. However, the UDP is limited to measuring velocities in a single direction and only the axial component or vertical component of the velocity in the WTHL test section can be obtained as a consequence of the test section geometry. While the primary advantage of the UDP technique is the ability to make meaningful time-dependent measurements, the velocities recorded may be averaged over time to establish typical conditions in the regions of the test section not visible in LDV studies, such as the region near the test section nose and the regions near each flow baffle.

Ultrasonic measurements were obtained at nine positions in the WTHL test section. All measurements were made along the horizontal center plane of the test section as shown in Figure 8.35. The orange dots on the nose of the test section in Figure 8.35 indicate the other UDP measurement positions. These measurements are separated by one-inch intervals and the fifth position coincides with the centerline of the exit channel. The time average of the velocities recorded at these positions by the UDP are shown in Figures 8.36 through 8.44. The error bars shown in these figures are not representative of the error in the measurement but of the standard deviation of the fluctuation in the measured velocity from the average value at that position. The actual error in the measurement is negligible in comparison to the fluctuation due to the turbulent nature of the flow field. Large fluctuations are recorded in the region near the wall of the test section because the interaction of

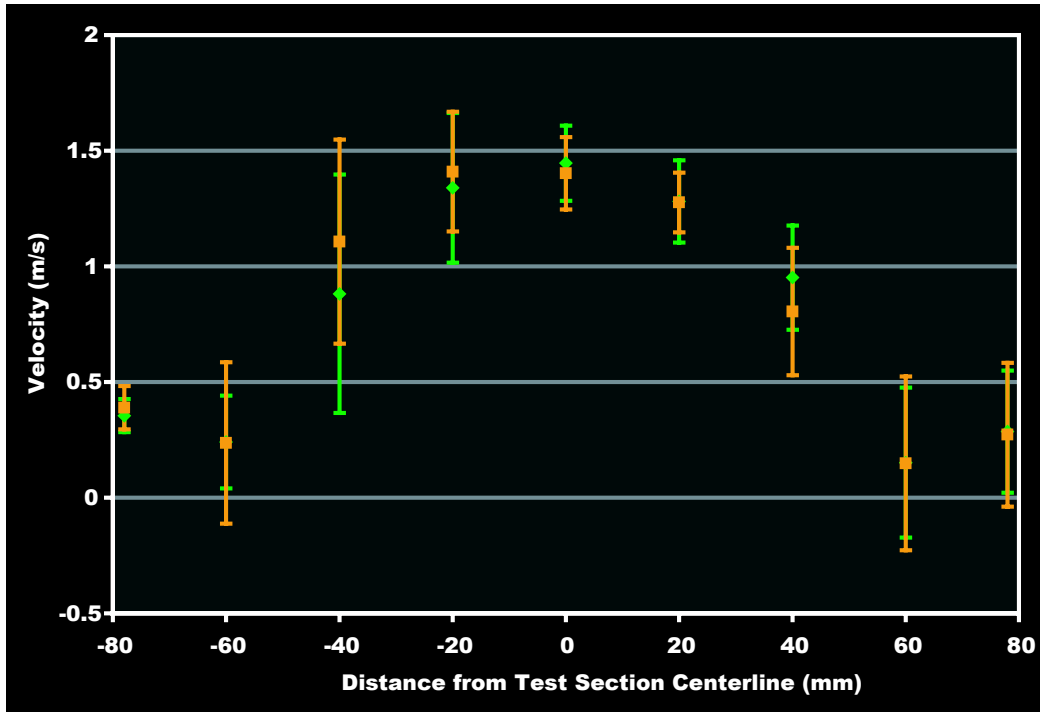


Figure 8.28 – Comparison of velocity measurements at horizontal midplane and a distance of 110.5 mm from the test section nose in nominal test section, with a flow rate of 5.5 L/s per inlet leg.

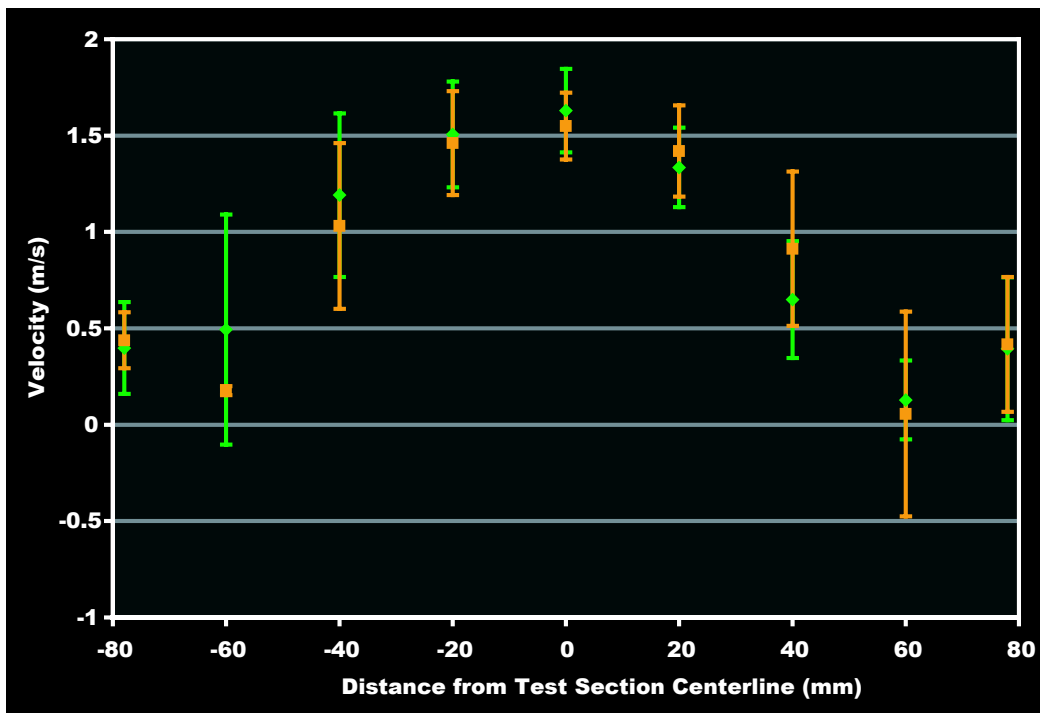


Figure 8.29 – Comparison of velocity measurements at horizontal midplane and a distance of 150.5 mm from the test section nose in nominal test section, with a flow rate of 5.5 L/s per inlet leg.

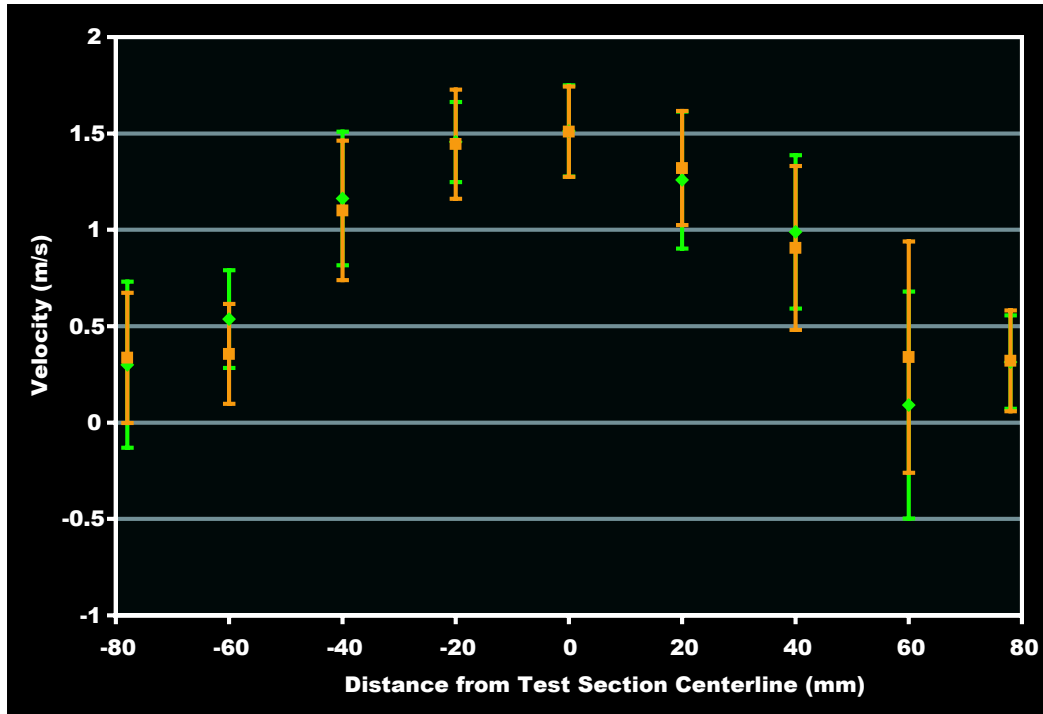


Figure 8.30 – Comparison of velocity measurements at horizontal midplane and a distance of 190.5 mm from the test section nose in nominal test section, with a flow rate of 5.5 L/s per inlet leg.

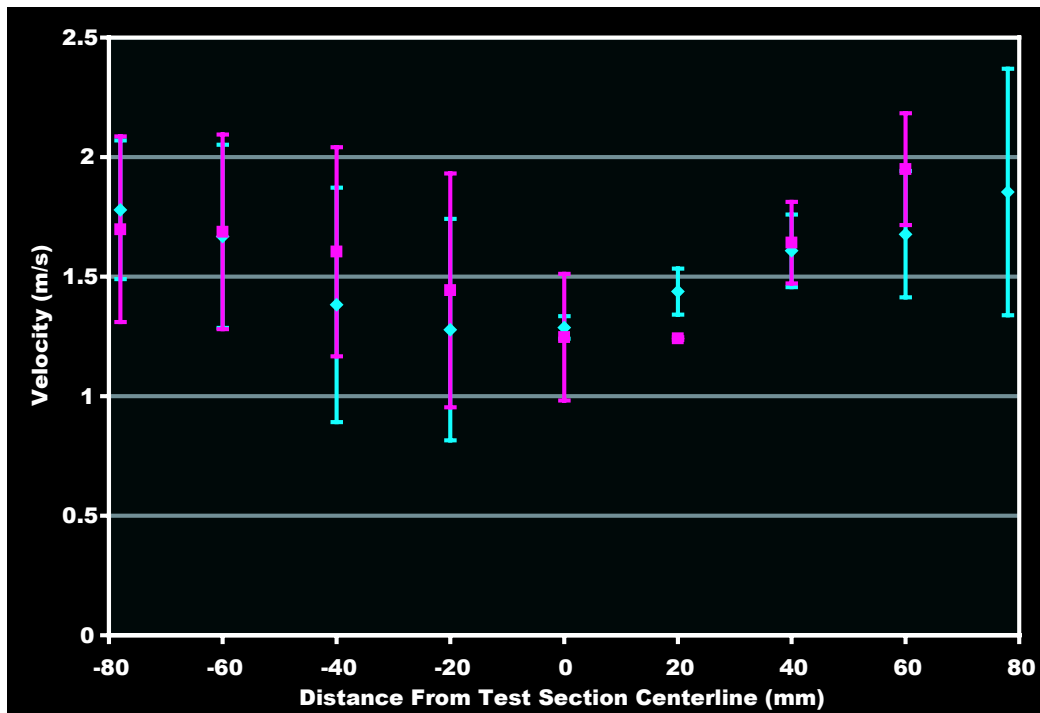


Figure 8.31 – Comparison of velocity measurements at horizontal midplane and a distance of 70.5 mm from the test section nose in nominal test section, with a flow rate of 7.7 L/s per inlet leg.

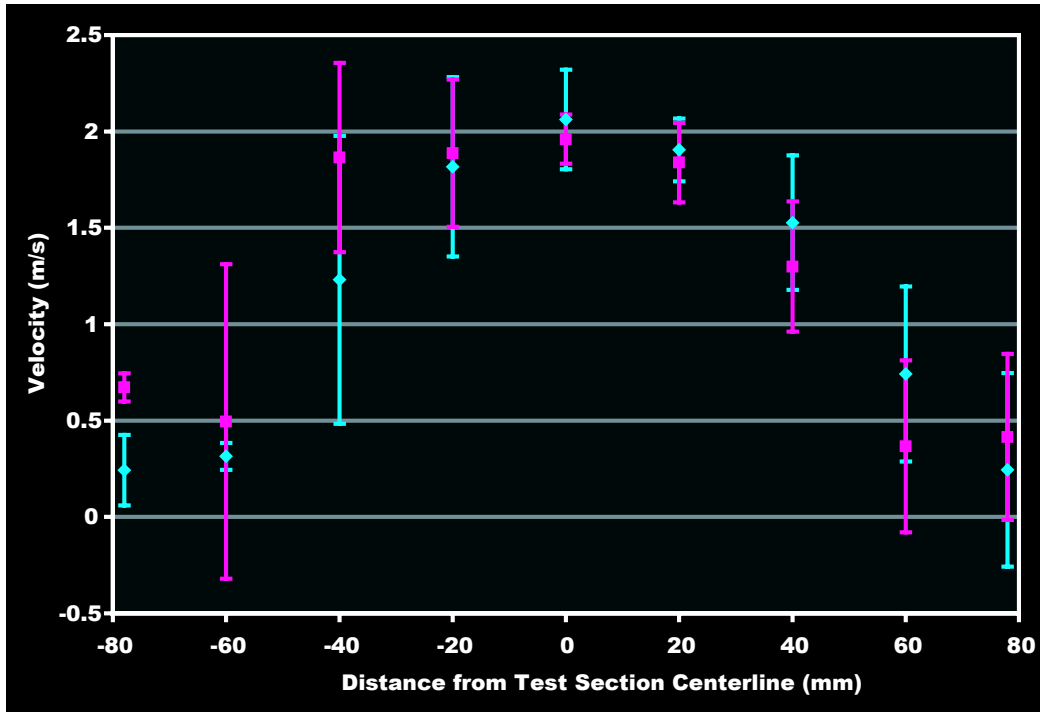


Figure 8.32 – Comparison of velocity measurements at horizontal midplane and a distance of 110.5 mm from the test section nose in nominal test section, with a flow rate of 7.7 L/s per inlet leg.

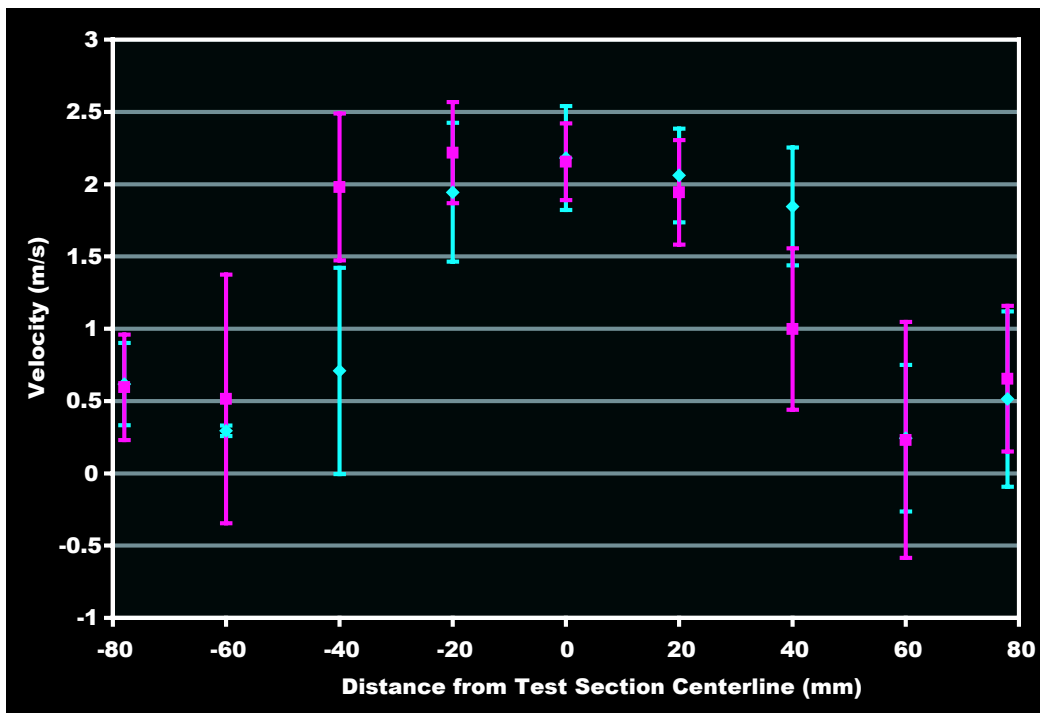


Figure 8.33 – Comparison of velocity measurements at horizontal midplane and a distance of 150.5 mm from the test section nose in nominal test section, with a flow rate of 7.7 L/s per inlet leg.

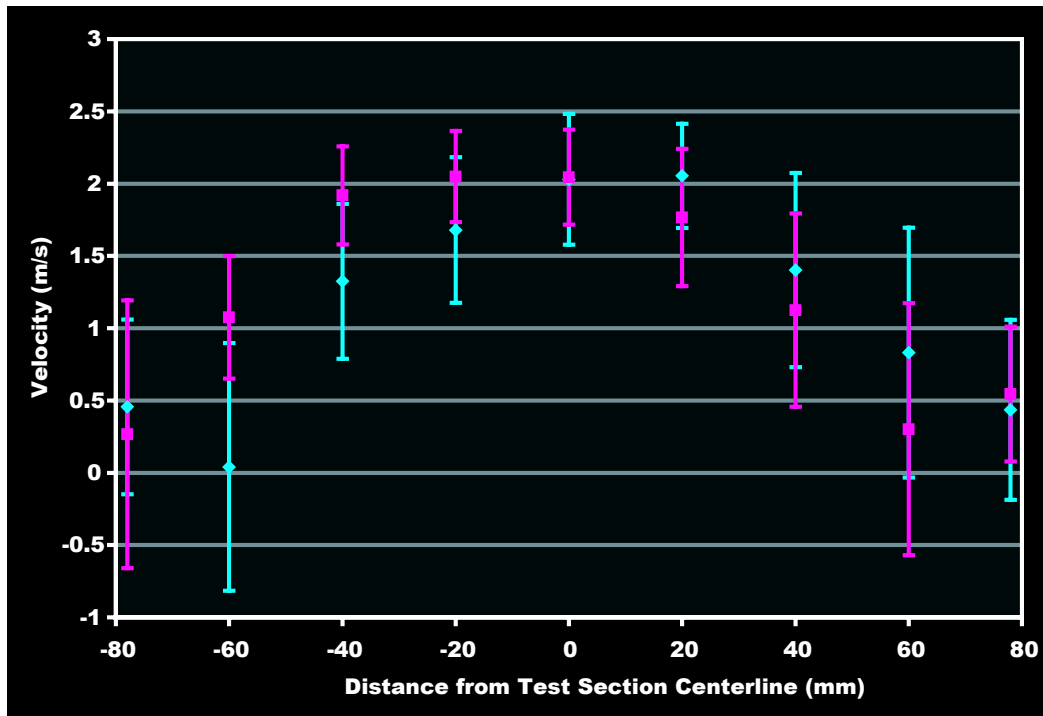


Figure 8.34 – Comparison of velocity measurements at horizontal midplane and a distance of 190.5 mm from the test section nose in nominal test section, with a flow rate of 7.7 L/s per inlet leg.

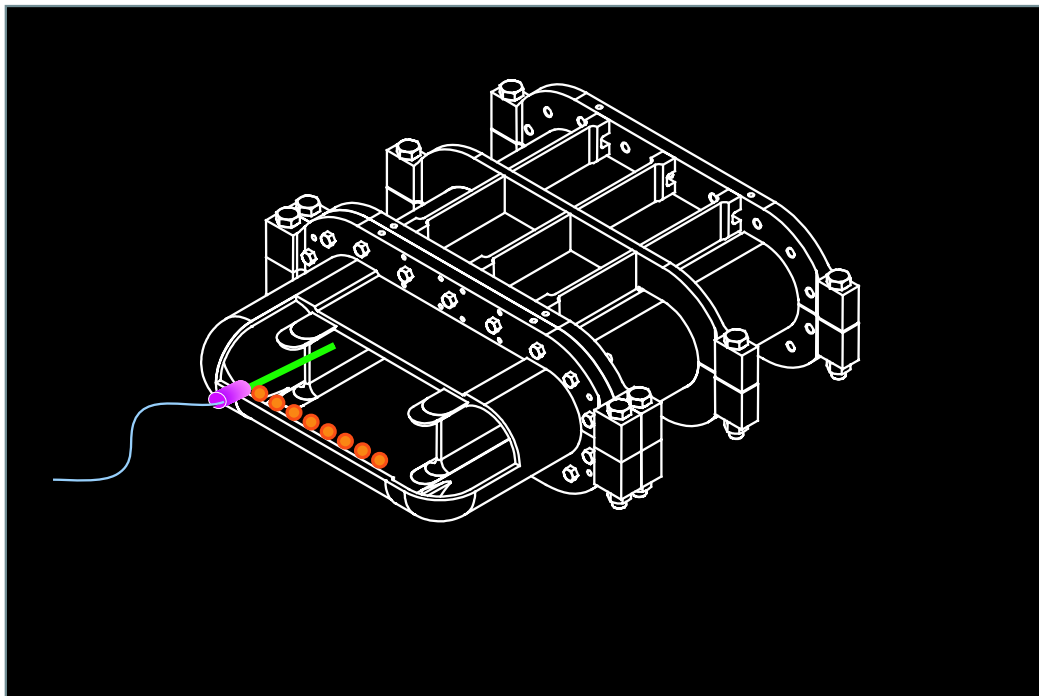


Figure 8.35 – Schematic showing typical position of the UVP probe.

the fluid and wall create reflected waves that limit the ability of the UVP to make accurate measurements in the first few millimeters of the fluid. In the regions near the flow baffle, negative velocities upstream of the leading edge of the flow baffle indicate that the flow has not yet turned to move down the exit channel. Negative velocities downstream of the leading edge of the flow baffle indicate that recirculation occurs along the baffle wall as predicted by the CFD simulation. In the regions near the centerline, the velocity is always positive as expected.

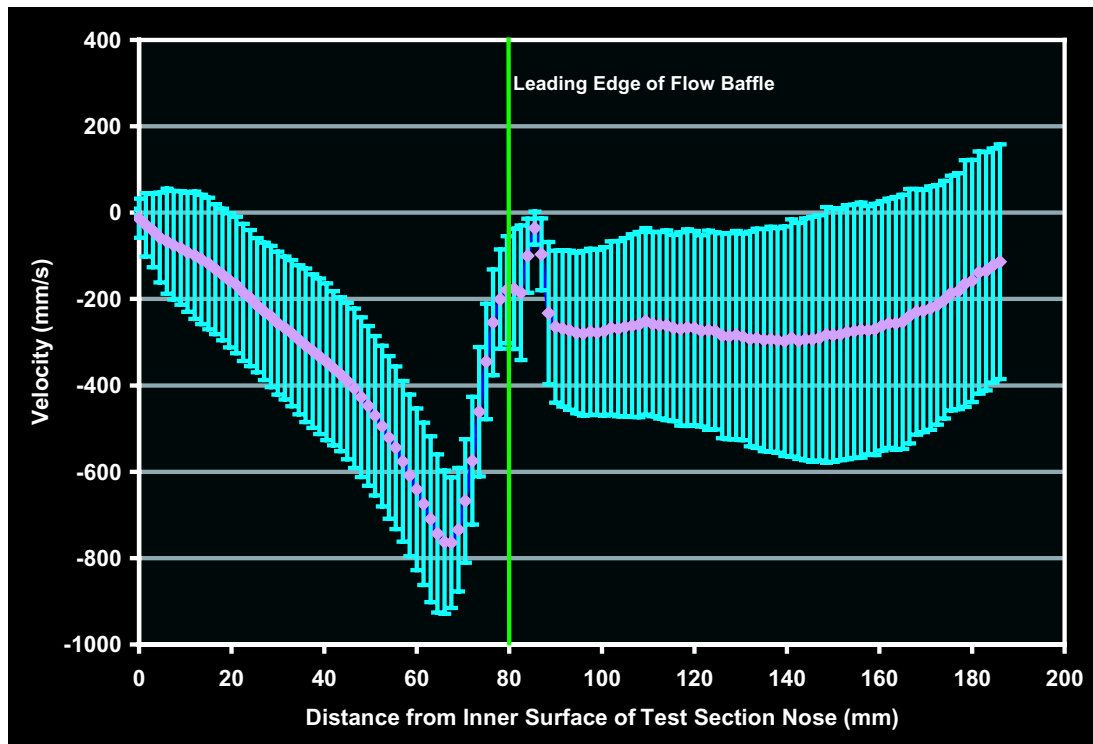


Figure 8.36 – Time-averaged velocity away from the test section nose at a distance of 4 inches to the right of the centerline of the exit channel.

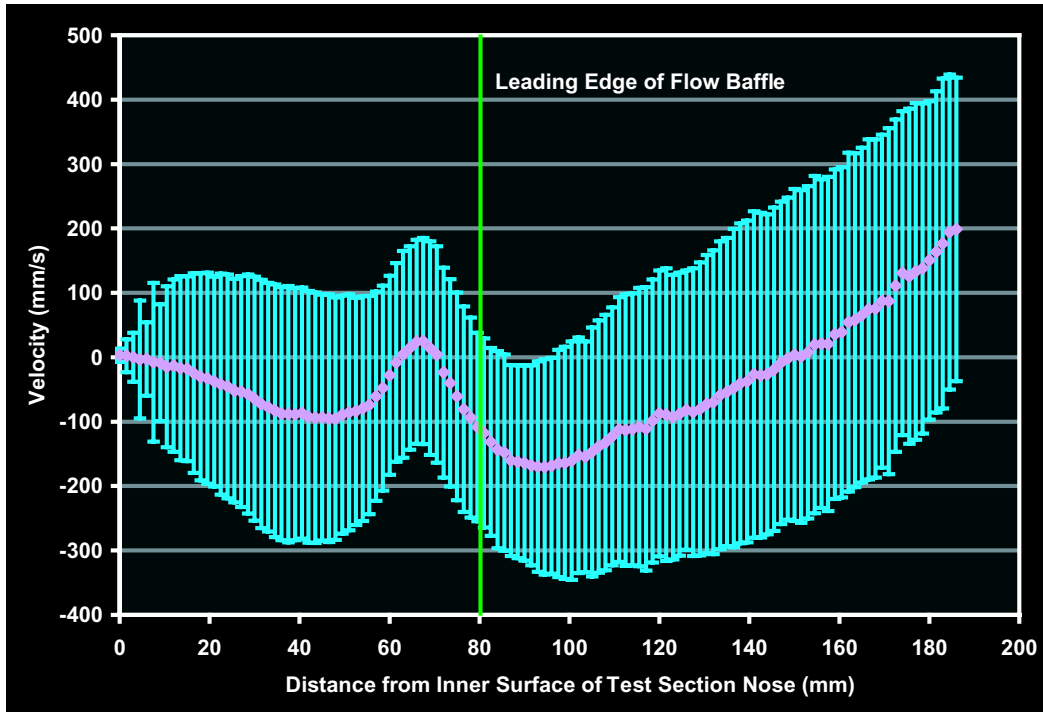


Figure 8.37 – Time-averaged velocity away from the test section nose at a distance of 3 inches to the right of the centerline of the exit channel.

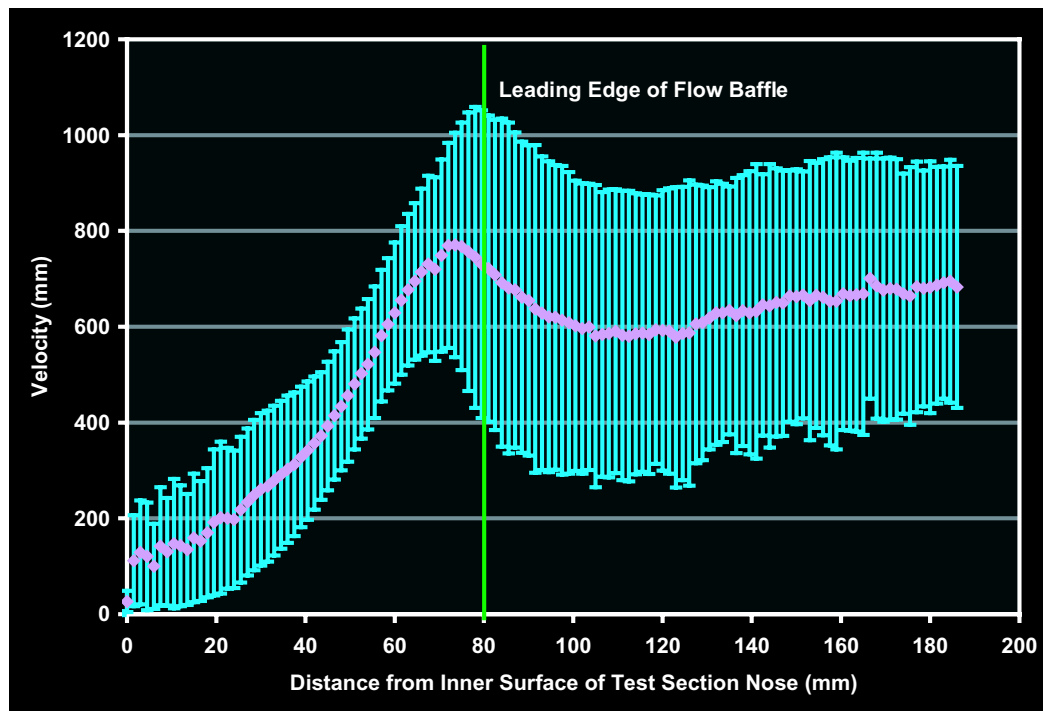


Figure 8.38 – Time-averaged velocity away from the test section nose at a distance of 2 inches to the right of the centerline of the exit channel.

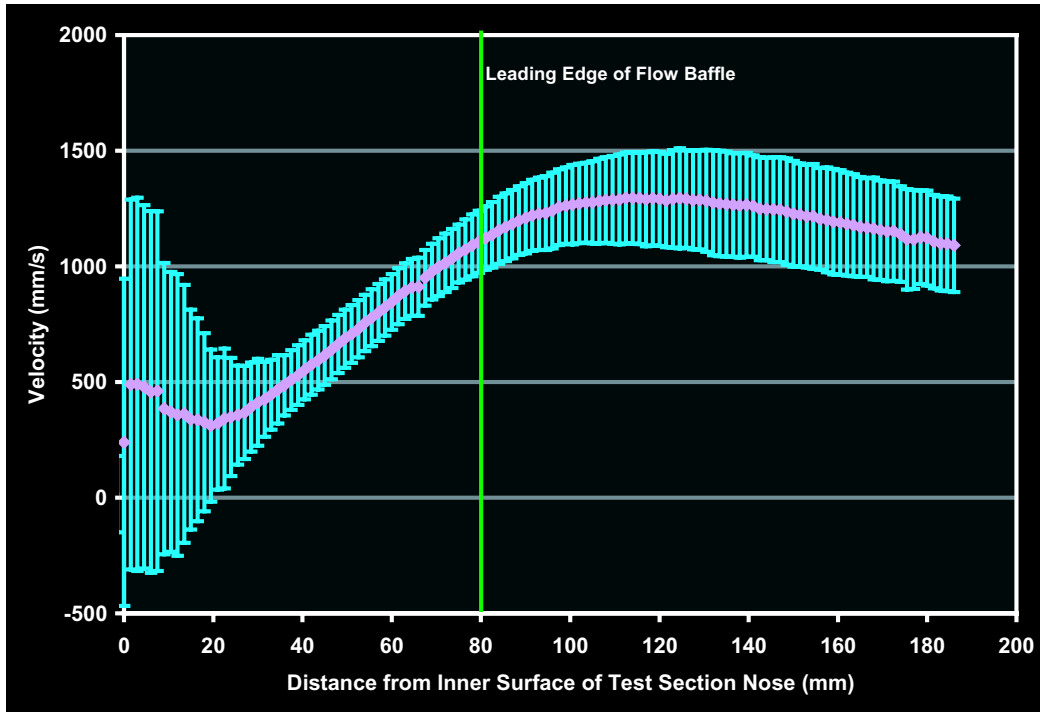


Figure 8.39 – Time-averaged velocity away from the test section nose at a distance of 1 inch to the right of the centerline of the exit channel.

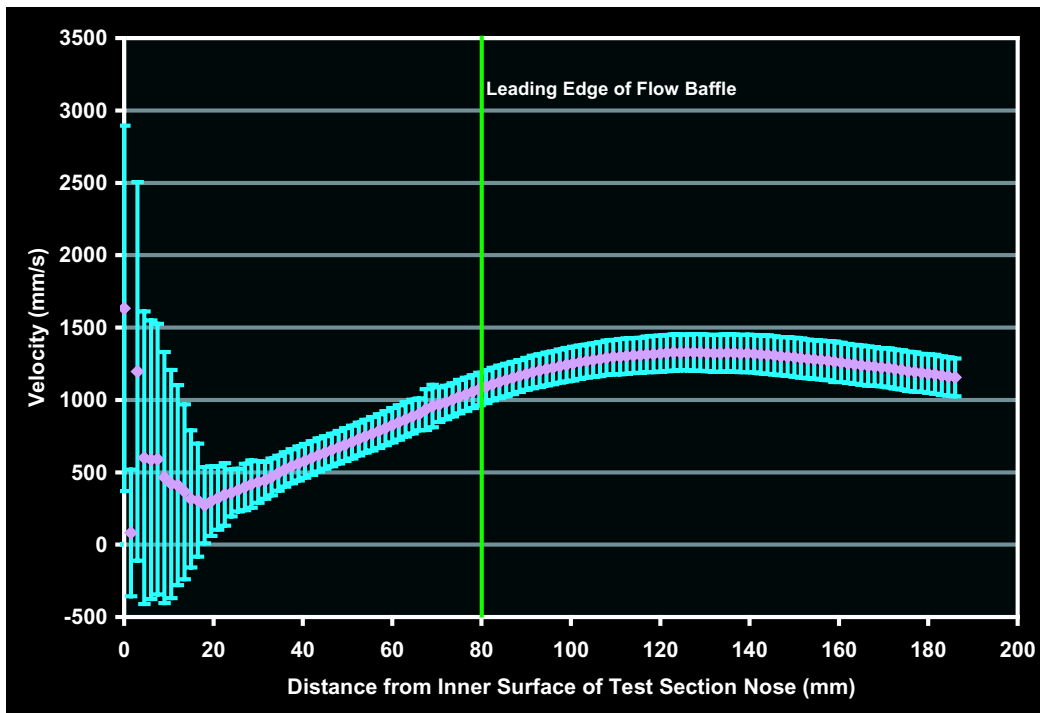


Figure 8.40 – Time-averaged velocity away from the test section nose at the centerline of the exit channel.

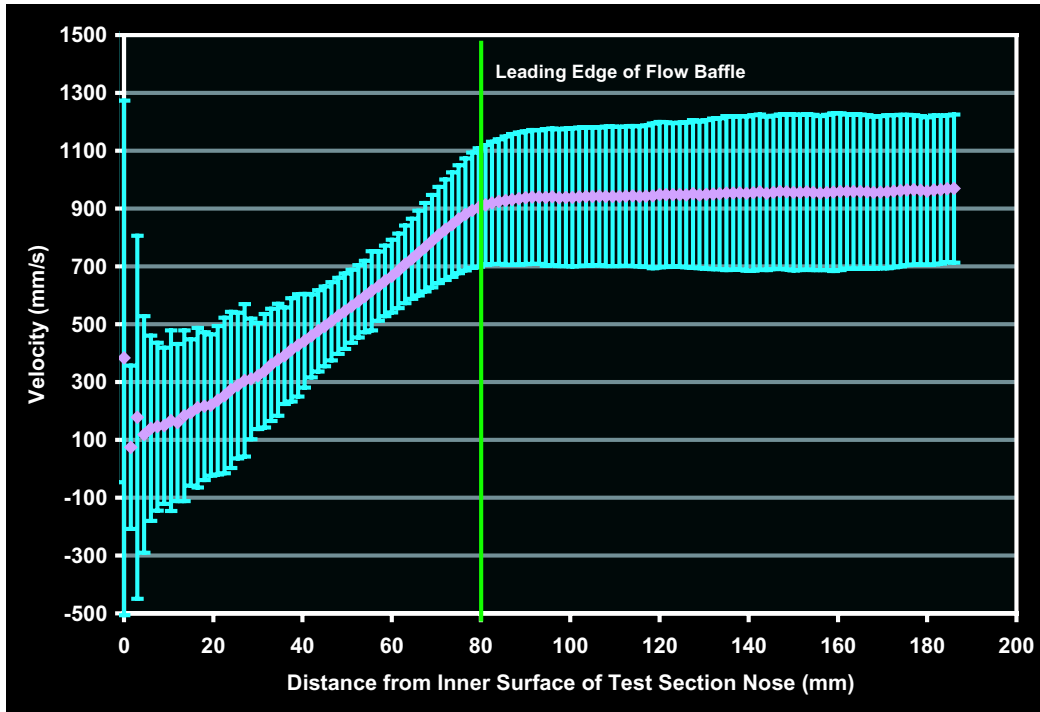


Figure 8.41 – Time-averaged velocity away from the test section nose at a distance of 1 inch to the left of the centerline of the exit channel.

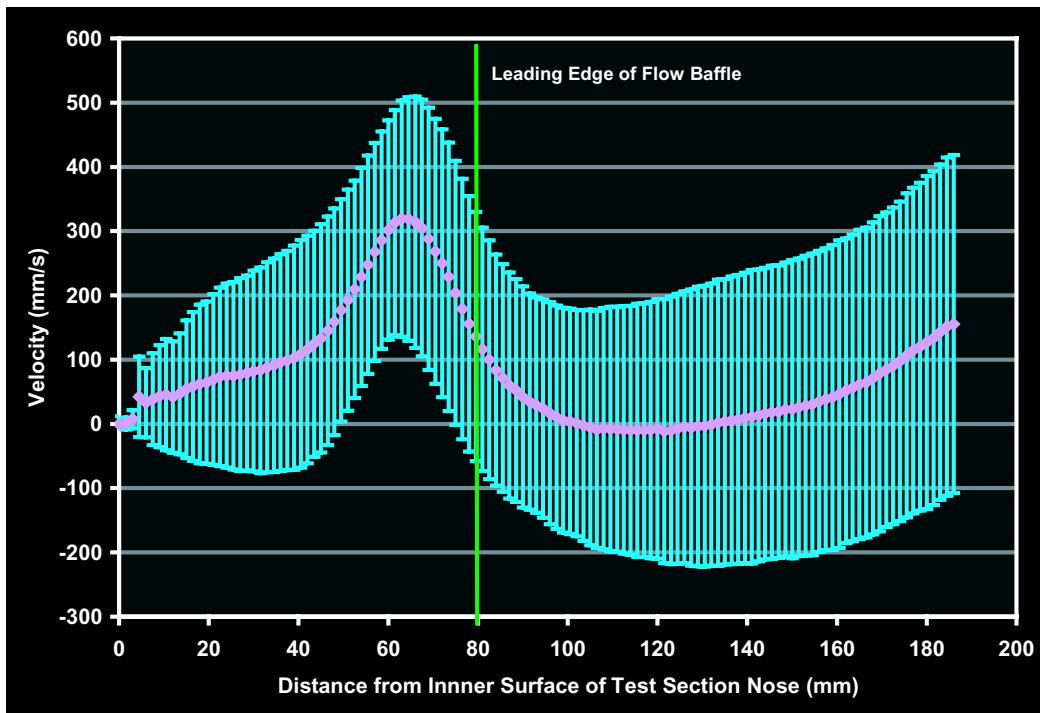


Figure 8.42 – Time-averaged velocity away from the test section nose at a distance of 2 inches to the left of the centerline of the exit channel.

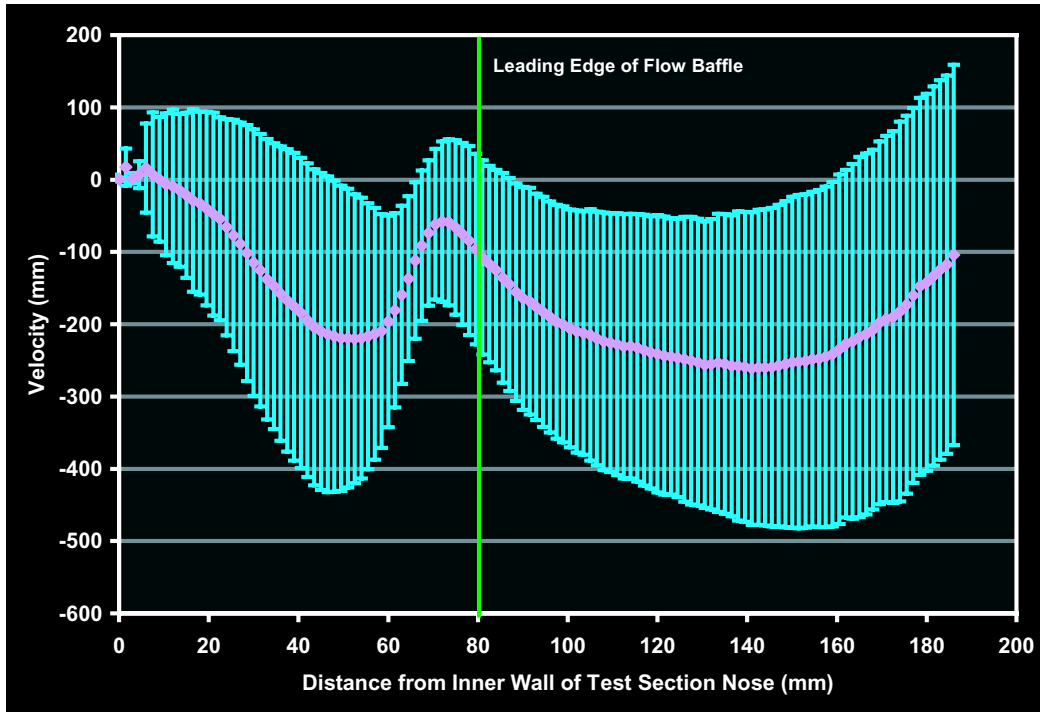


Figure 8.43 – Time-averaged velocity away from the test section nose at a distance of 3 inches to the left of the centerline of the exit channel.

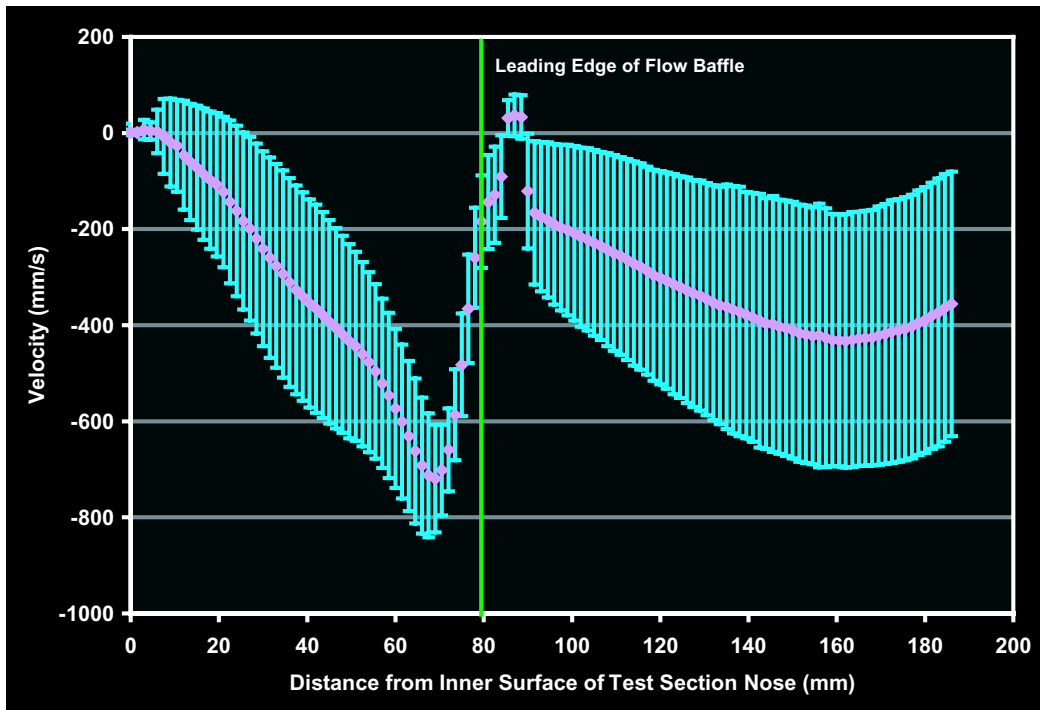


Figure 8.44 – Time-averaged velocity away from the test section nose at a distance of 4 inches to the left of the centerline of the exit channel.

Chapter 9 Evaluation of Potential Flow Control Devices in the WTHL

Since the Computational Fluid Dynamics (CFD) simulation will be the primary thermal hydraulic tool for the development of modifications in future targets, the robustness of the computational model for extrapolation to new flow geometries must be established. Initial CFD studies indicate that slight modifications in the target geometry could provide a more stable flow field and encourage better heat transfer along the exit channel surface of the two flow baffles. Two modifications were made to the Water Thermal Hydraulic Loop (WTHL) test section to provide data for the validation of the CFD simulation. First, the two flow baffles that separate the inlet and exit channels in the test section were extended by 2.254 cm, reducing the flow area between the leading edge of the flow baffle and the front of the beam window by 25 percent. Additionally a flow vane was installed around the leading edge of the flow baffle to encourage flow along the exit channel side of the baffle where recirculation zones develop in the nominal test section. The effects of these alterations in the test section as predicted by the CFD model are shown in Figure 9.1. The CFD model indicates that these modifications lead to the development of a significant flow path between the baffle wall and the stationary recirculation zone near the leading edge of the flow baffle. The existence of such a path would significantly reduce the heat load in the structure of the flow baffle.

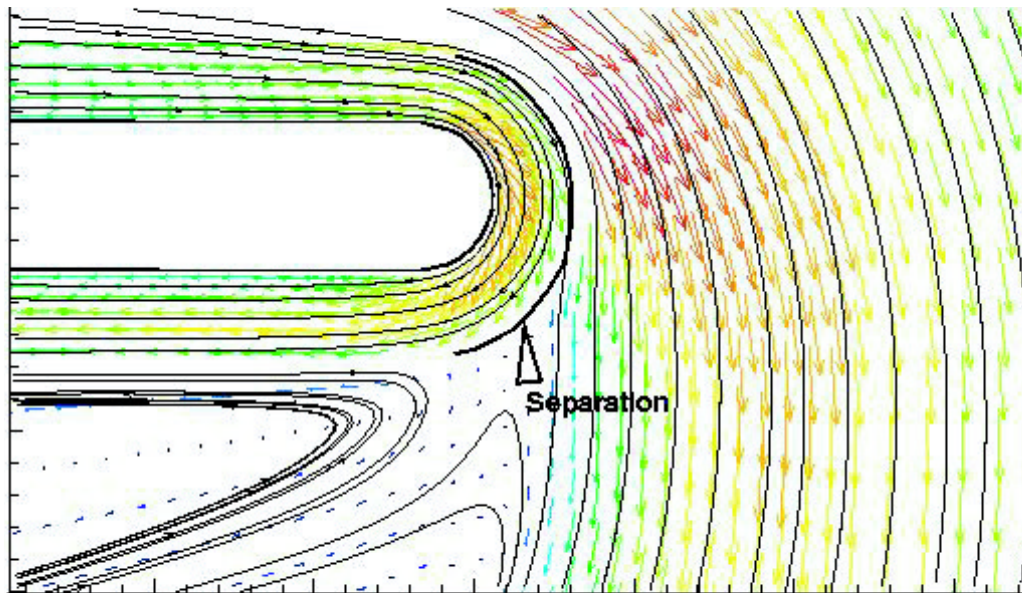


Figure 9.1 – Close up view of region surrounding the leading edge of the flow baffle from results of a CFD simulation of the flow field in the modified WTHL test section (Wendel, 2001 – used with permission).

9.1 Flow Visualization Studies in the Modified WHTL Test Section

Flow visualization studies provide the opportunity to visually observe the behavior of the flow field and identify regions of interest for further study. While the results of these studies are qualitative rather than quantitative, they are useful for the identification of flow behaviors that are unexpected. Two flow visualization methods are used in the WTHL experiments: dye injection and bubble injection. Bubbles and dye are injected into the test section at one of two positions in these studies. The first position is along the surface of the flow baffle near the inlet to the flow vane. The second position is along the outer surface of the flow vane. Flow visualization studies are performed at both the nominal WTHL flow rate (11 l/s per inlet leg) and half of the nominal WTHL flow rate (5.5 l/s per inlet leg). Images from dye visualization studies in the modified test section at half of the nominal flow rate are shown in Figures 9.2 and 9.3.

Flow visualization studies indicate that the modifications to the WTHL test section do not drastically alter the flow behavior in the test section. However there are some notable differences in the flow behavior seen in the modified and nominal test sections. In the modified test section, flow separation occurs along the flow vane rather than at the leading edge



Figure 9.2 – Flow visualization study in the modified WTHL test section with dye injected along the inside surface of the flow vane. The flow rate is half of the nominal flow rate (5.5 l/s per inlet leg). To see video footage of this experiment, click on the image to activate the animation.



Figure 9.3 – Flow visualization study in the modified WTHL test section with dye injected along the outside surface of the flow vane. The flow rate is half of the nominal flow rate (5.5 l/s per inlet leg). To see video footage of this experiment, click on the image to activate the animation.

of the flow baffle. The repositioning of the separation point results in a slight increase in the size of the primary recirculation zones that form near the leading edge of each flow baffle. Furthermore, the fluid that flows between the flow vane and the baffle appears to continue to travel between the baffle wall and the primary recirculation zone for a short distance after it leaves the flow vane in the exit channel. In the half-nominal flow cases, a secondary recirculation zone clearly develops from this flow between the baffle wall and the primary recirculation zone approximately 3 cm from the leading edge of the flow baffle. In the nominal flow cases, it is not clear whether the flow along the baffle from the flow vane develops a secondary recirculation zone or is simply absorbed by the primary recirculation zone at some position. Finally, continuous large-scale turbulent behavior is observed in the modified test section rather than the pseudo-periodic vortex shedding that was observed in the nominal test section. This behavior may be responsible for a noticeable increase in the difficulty associated with maintaining flow rates through the two inlets with fluctuations less than 1 percent of the total flow rate.

9.2 Evaluation of the Pressure Field in the Modified WHTL Test Section

As with the nominal WTHL test section the pressure drops in the nominal test section was evaluated using 4 pressure transducers placed in the same locations as in the previous study in the nominal test section. The pressure drop in the test section was evaluated for three different flow rates, 11, 7.14, and 12.75 L/s per inlet leg. Results are shown in Table 9.1 for each pressure measurement and the calculated pressure drop. The reported error is the observed variation in the measurement resulting from turbulent flow behavior. The variation in the flow rate in these experiments is approximately 2 percent.

Table 9.1 – Pressure drop data for modified test section. Time-dependent variations are reported as percent for flow rate measurements and psig for pressure measurements.

Flow Rate (l/s) Per Inlet Leg	PT-1 (psig)	PT-2 (psig)	PT-3 (psig)	PT-4 (psig)	PT-5 (psig)	Average Test Section Pressure Drop (psi)
11.0 +/- 2%	N/A	1.44 +/- 0.2	1.39 +/- 0.15	1.07 +/- 0.20	13.0 +/- 0.75	0.37
7.14 +/- 2%	N/A	0.84 +/- 0.1	0.74 +/- 0.1	0.72 +/- 0.07	10.25 +/- 0.25	0.12
12.75 +/- 2%	N/A	1.74 +/- 0.25	1.74 +/- 0.25	1.27 +/- 0.15	20.0 +/- 2.0	0.47

9.3 Evaluation of Time-Averaged Characteristics of the Velocity Field in the Modified WTHL Test Section

To provide qualitative data for validation of the CFD simulations, flow velocity measurements are made at various positions in the WTHL test section. These measurements are made using a two-dimensional Laser Doppler Velocimeter that simultaneously measures two components of the flow velocity at a single point in the flow field. Velocity mapping is accomplished using a traversing system that positions the measurement volume to set locations within the test section. The LDV system records as many as four thousand individual velocity measurements at each position in the test section, and the system software reports the average value of those measurements.

LDV measurements in the modified WTHL test section are limited to the region visible through a single acrylic window that is machined into the plastic material from which the majority of the test section is manufactured. The window is necessary because the optical quality of the plastic material is not sufficient to allow quality LDV measurements. Within the region visible through the acrylic window, the flow velocity is mapped at five depths in the flow field:

1. 10 mm below the upper inside surface of the test section
2. 20 mm below the upper inside surface of the test section
3. At the horizontal midplane of the test section
4. 20 mm above the lower inside surface of the test section
5. 10 mm above the lower inside surface of the test section

At each depth, a total of 63 point velocity measurements are recorded. Results of the velocity mapping are shown in Figures 9.4 through 9.8. In addition to mapping the velocity field at each depth, the turbulence intensity is calculated for each position in the test section. The turbulence intensity maps for each depth in the test section are shown in Figures 9.9 through 9.13.

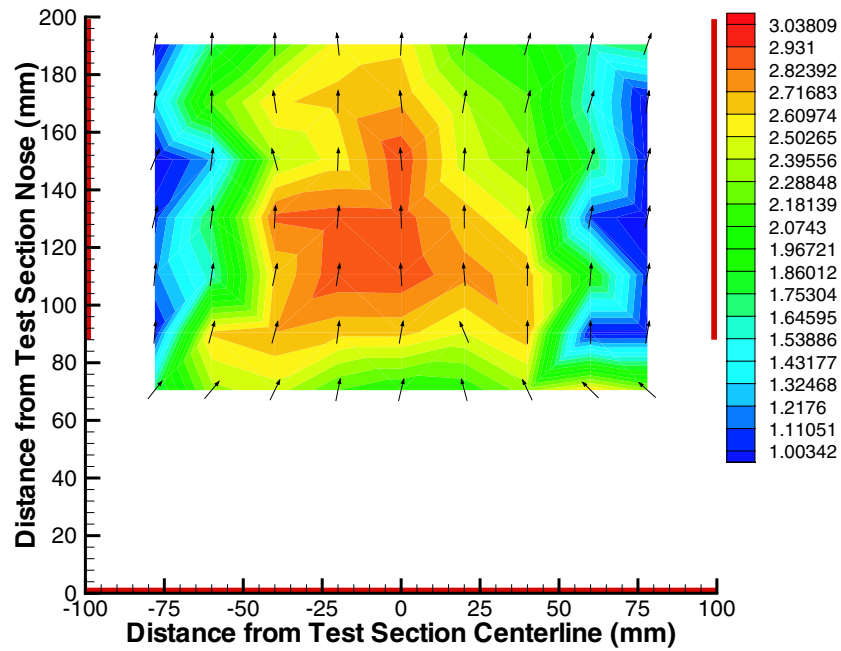


Figure 9.4 – Velocity in modified test section at a depth of 10 mm from the upper inside surface.

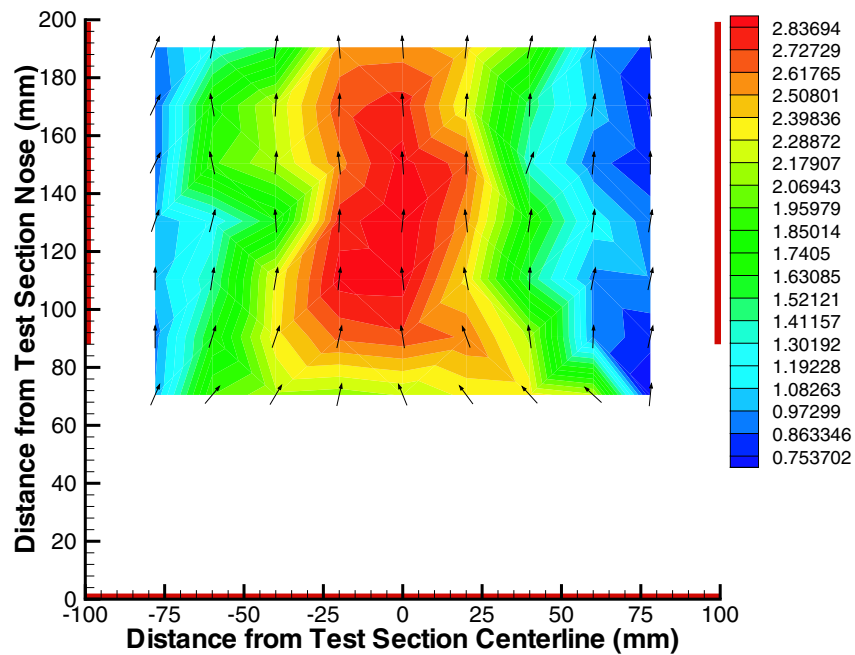


Figure 9.5 – Velocity in modified test section at a depth of 20 mm from the upper inside surface.

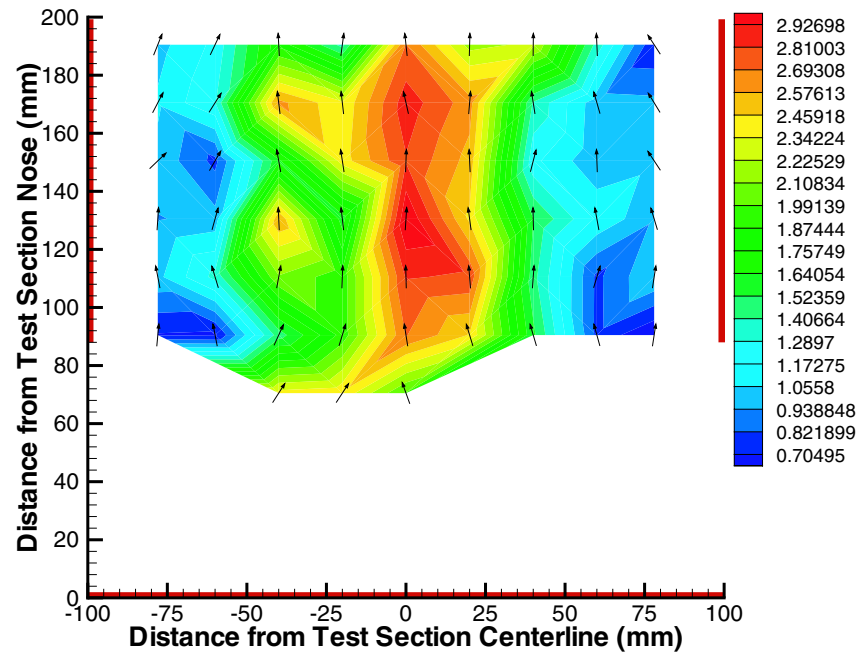


Figure 9.6 – Velocity in modified test section at the horizontal mid-plane.

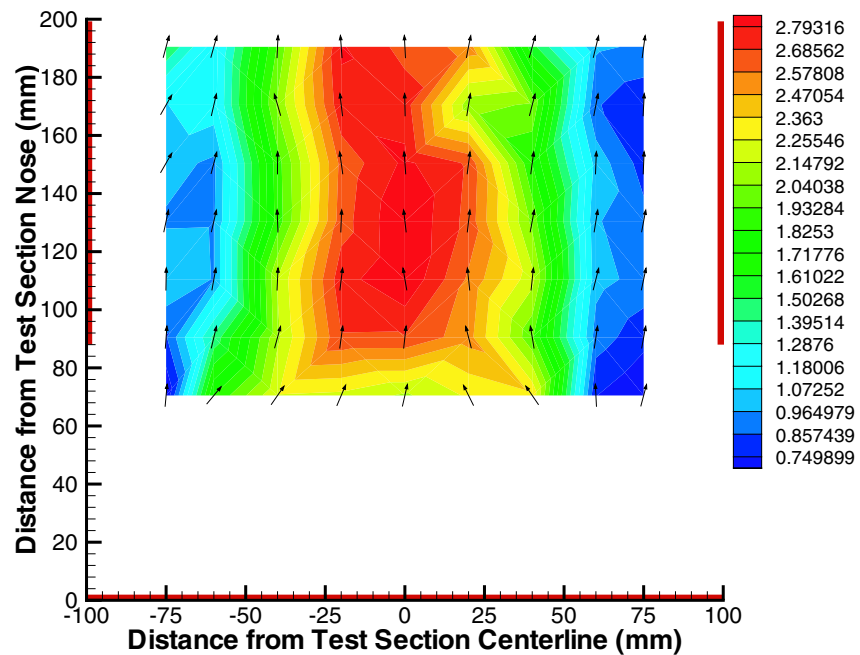


Figure 9.7 – Velocity in the modified test section at a depth of 20 mm from the lower inside surface.

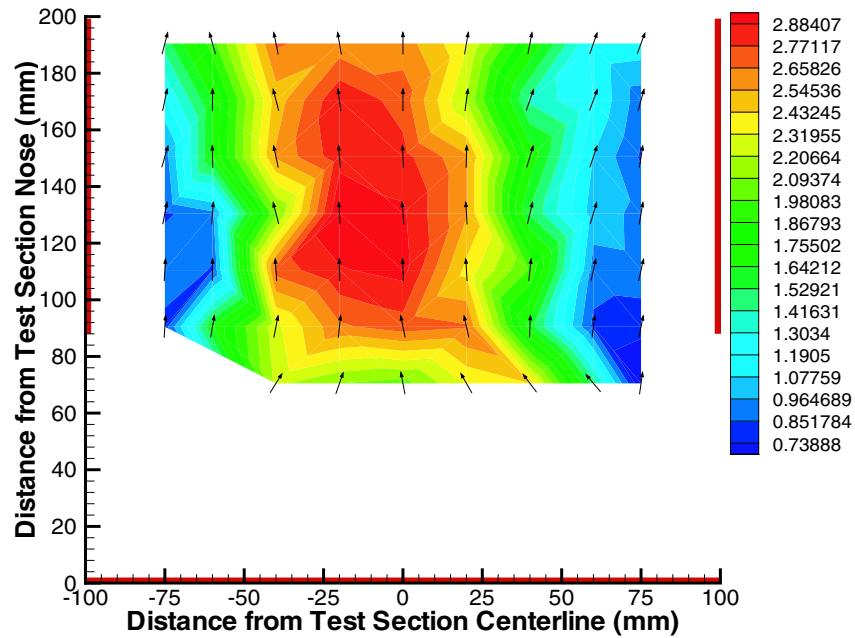


Figure 9.8 – Velocity in the modified test section at a depth of 10 mm from the lower inside surface.

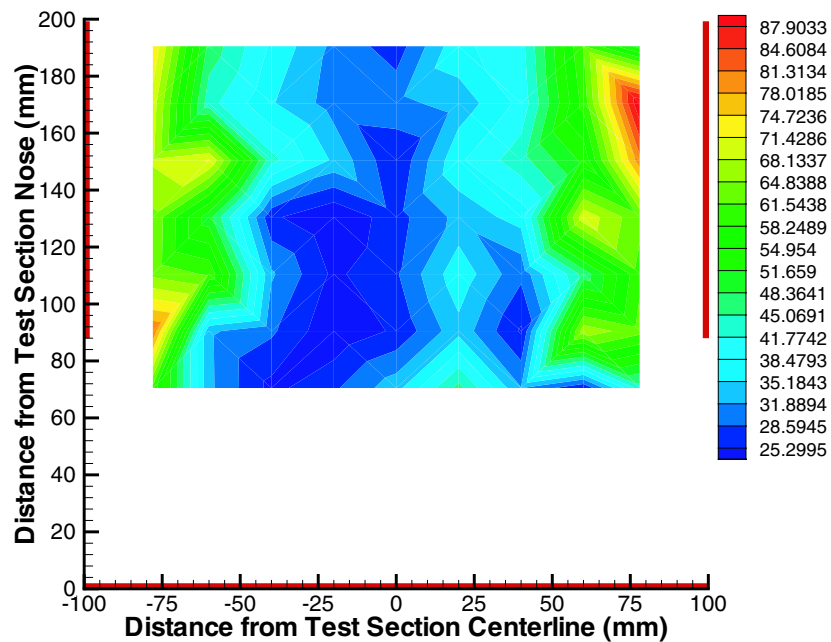


Figure 9.9 – Turbulence intensity in the modified test section at a depth of 10 mm from the upper inside surface.

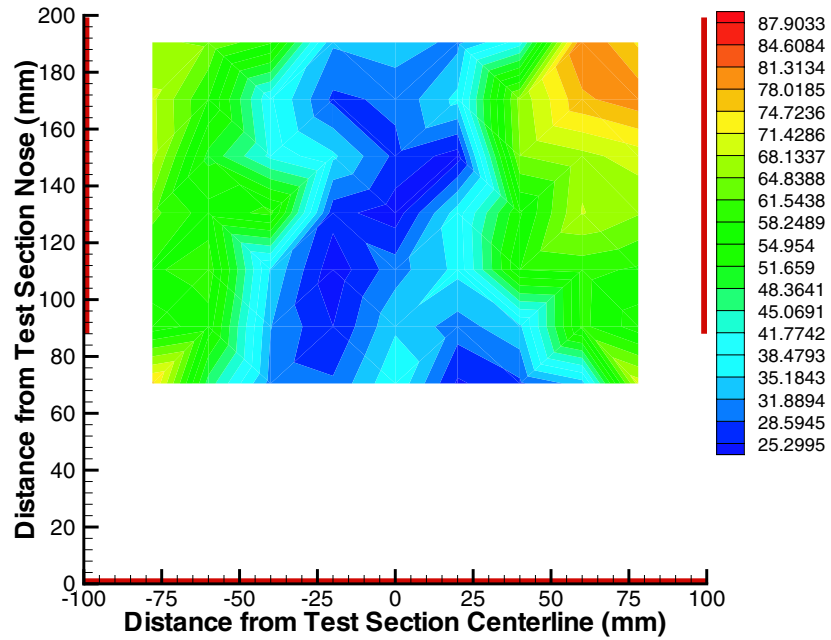


Figure 9.10 – Turbulence intensity in the modified test section at a depth of 20 mm from the upper inside surface.

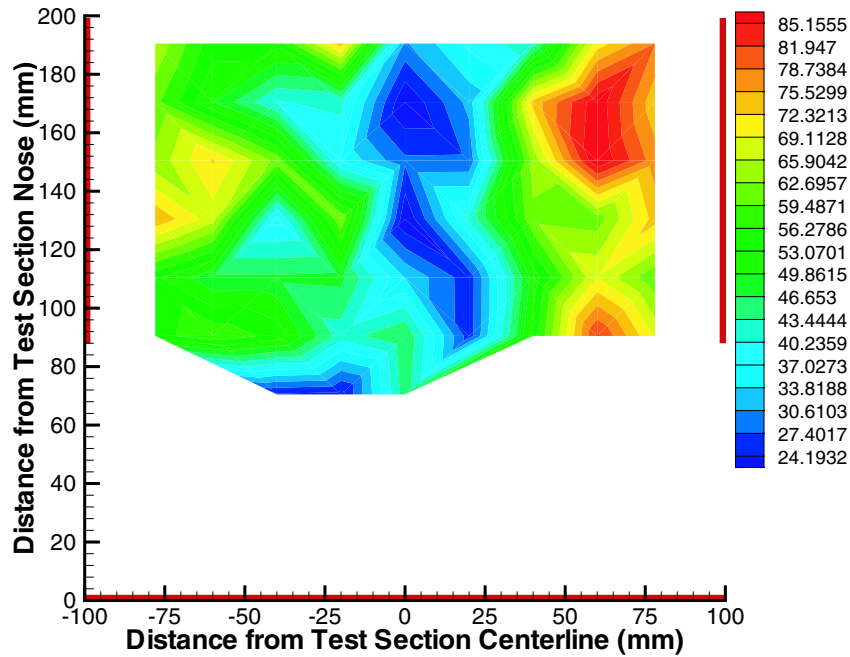


Figure 9.11 – Turbulence intensity in the modified test section at the horizontal mid-plane.

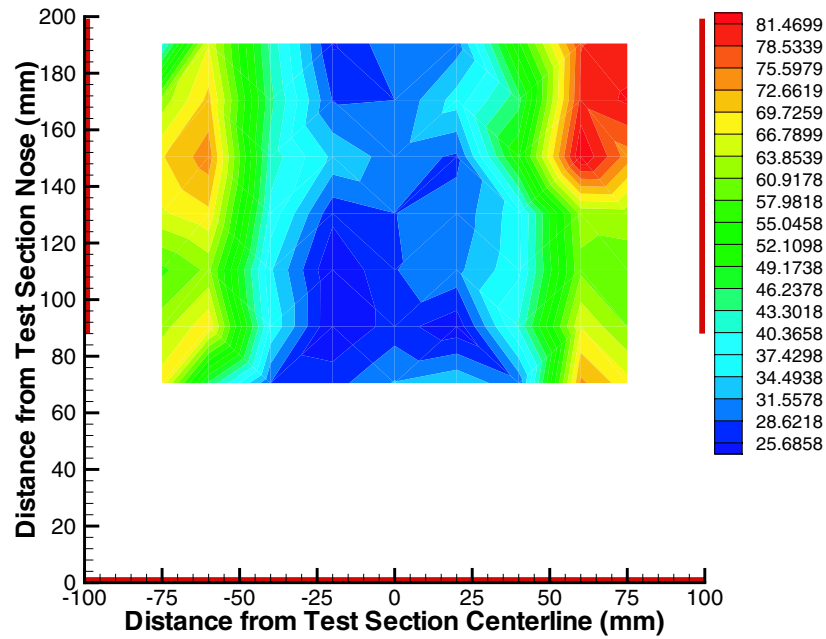


Figure 9.12 – Turbulence intensity in the modified test section at a depth of 20 mm from the lower inside surface.

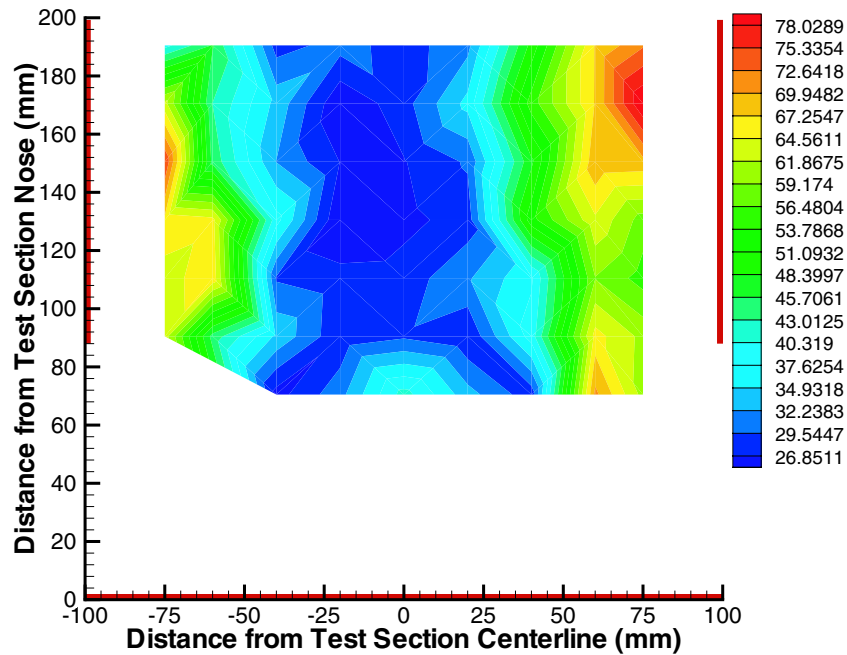


Figure 9.13 – Turbulence intensity in the modified test section at a depth of 10 mm from the lower inside surface.

9.4 Repeatability of Velocity Measurements in the Modified WTHL Test Section

As in the previous studies with the nominal test section, multiple data sets were recorded at the horizontal mid-plane of the test section in order to evaluate the repeatability of the experiment. The measured velocities are plotted together for comparison in Figures 9.14 through 9.16. Each figure indicates the measured total velocity as a function of distance from the vertical midplane at one of three distances (110.5, 150.5, and 190.5 mm) from the test section nose. The error bars indicate the standard deviation in the velocity measurement at each measurement position. The standard deviation is indicative only of the fluctuation during the period of measurement at each point, not the error associated with each measurement. The error associated with the measurement itself is negligible in comparison to these fluctuations. While the error associated with the motion of the traverse system is negligible, the spatial error resulting from the orientation of the test section and the traversing system is approximately ± 2 mm.

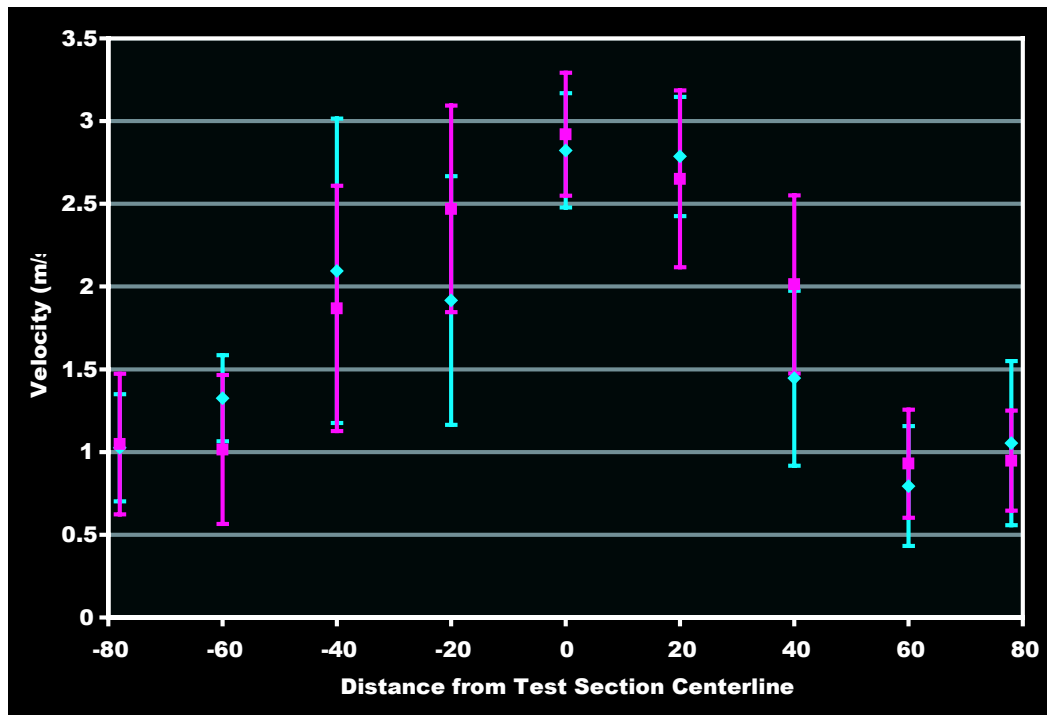


Figure 9.14 – Velocity comparison at a distance of 110.5 mm from test section nose.

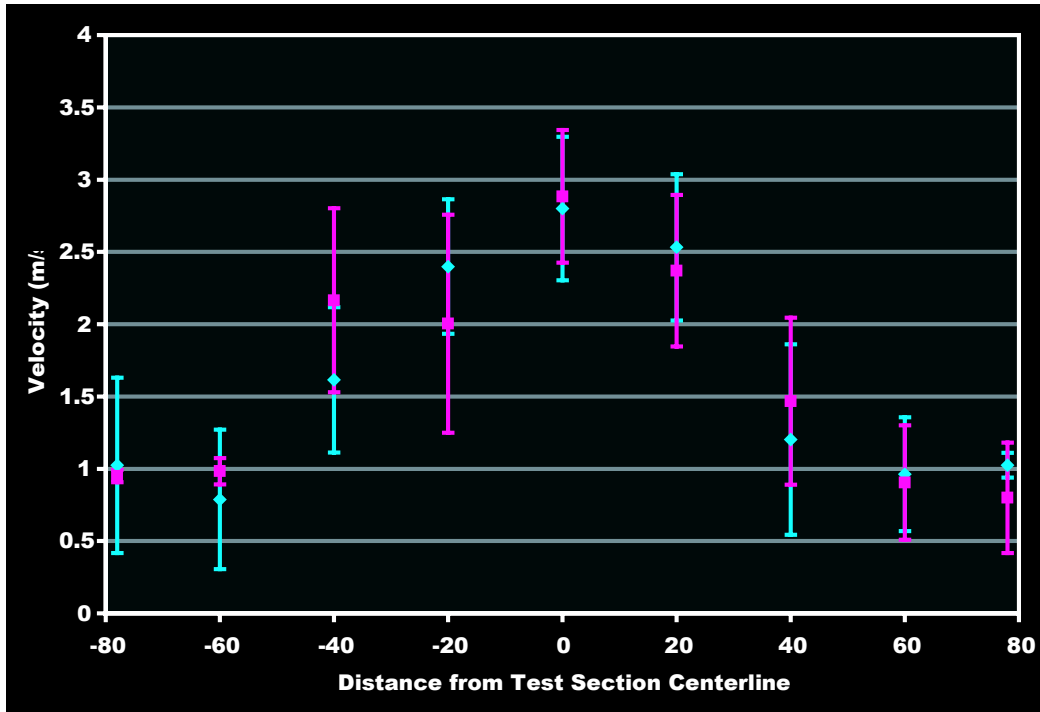


Figure 9.15 – Velocity comparison at a distance of 150.5 mm from test section nose.

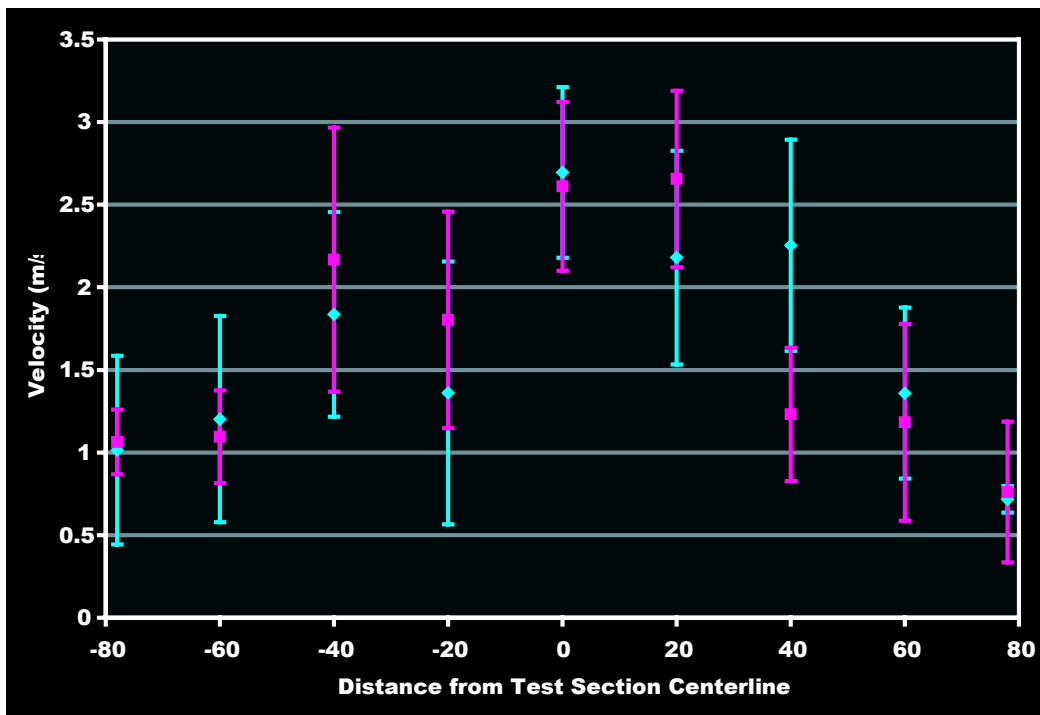


Figure 9.16 – Velocity comparison at a distance of 190.5 mm from test section nose.

9.5 Comparisons of Velocity Mappings in Nominal and Modified WTHL Test Sections

The flow visualization studies indicate that the modifications to the test section result in the development of an additional pair of stationary recirculation zones in the exit channel of the WTHL test section. These also appear to indicate an increase in large-scale turbulence patterns in the exit channel of the test section. Additionally, the flow does not appear to be as diffusive in the modified test section since the dye is not rapidly spread across the width of the exit channel.

The velocity maps developed in both the nominal and modified test sections are compared in Figure 9.17, and the turbulence intensity maps are compared in Figure 9.18. As indicated by the comparison, the addition of the turning vanes leads to a decrease in the velocity of the fluid in the exit channel. The turbulence intensity in the modified test section is also significantly higher than in the nominal case. These effects would tend to have a negative impact on the heat transfer coefficient between the exit channel flow and the flow baffle. Furthermore, the

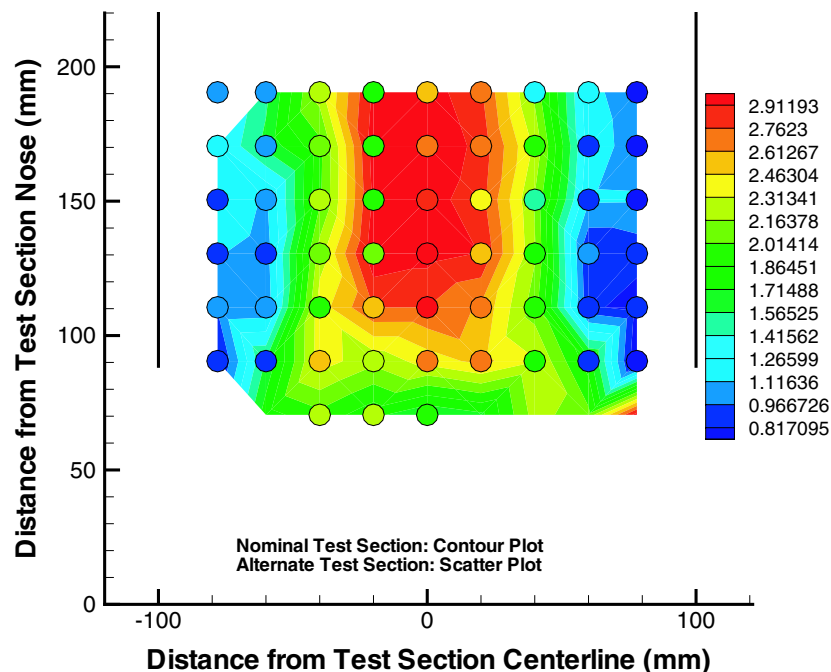


Figure 9.17 – Comparison of LDV velocity data in the nominal and modified WTHL test section. The data from the nominal test section is shown as a contour plot, and the data from the modified test section is shown as individual data points.

addition of the flow vanes does not appear to provide a significant stable flow of cooler fluid between the flow baffle and the primary recirculation zone as predicted by the CFD. Indeed, additional CFD studies with alternative turbulence models and grid structures indicate that the flow along the baffle may not develop. Therefore, the addition of turning vanes provides little benefit to the design of the SNS mercury target.

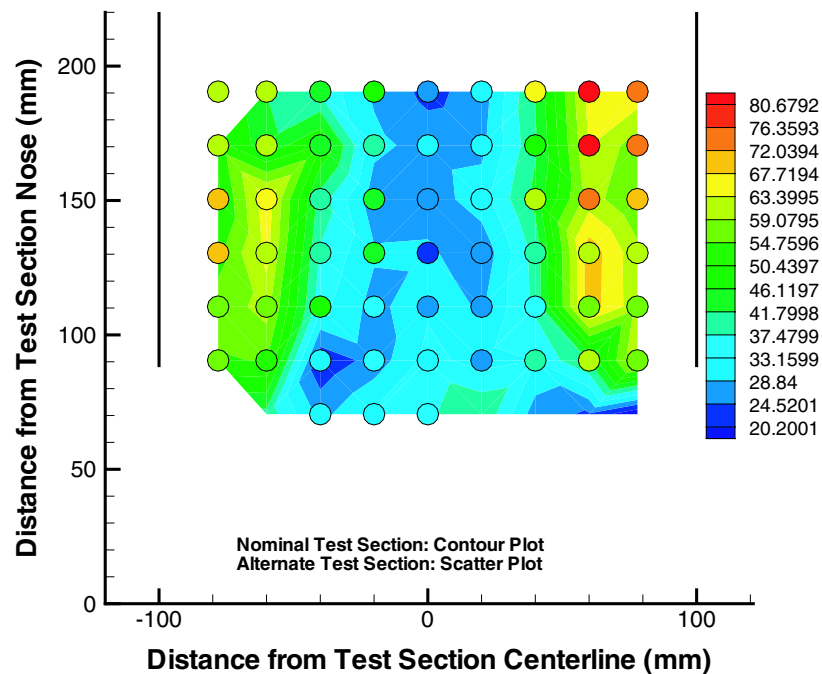


Figure 9.18 – Comparison of LDV turbulence intensity data in the nominal and modified WTHL test section. The data from the nominal test section is shown

Chapter 10 Evaluation of Time-Dependent Characteristics of the Velocity Field in the SNS Thermal Hydraulic Experiments

The primary fluid flow phenomena that could impact the expected lifetime of the Spallation Neutron Source (SNS) mercury target, as identified by the Computational Fluid Dynamics (CFD), are the formation of the two stationary recirculation zones at the leading edge of each flow baffle and the formation of a third recirculation zone in the nose region of the target. Under certain conditions the existence of these recirculation zones could lead to the development of high temperatures or large temperature gradients in the structure of the target. The flow visualization studies, discussed in Chapter 6, indicate that the recirculation zone in the nose region of the target is highly unstable and, as a result, will have little effect on the temperature distributions in that region. These studies indicate that the larger recirculation zones at the leading edge are relatively stable and that they will exist in the exit channel flow field throughout the operation of the facility. Furthermore, velocity mapping experiments indicate that these recirculation zones exhibit significantly lower flow velocities than the remainder of the flow field. If the residence time of a fluid element in the recirculation zone is sufficiently long that the fluid element is exposed to multiple proton pulses, temperatures of the fluid and the surrounding structure could be elevated above acceptable levels. Since the recirculation zones are slow moving, the turbulence of the flow field must be relied upon to prevent the temperature of the fluid from reaching undesirable levels. In order to evaluate the likelihood of substantial turbulent dissipation and diffusion in these zones, the time-dependent behavior of the flow velocity in these regions was examined in the SNS thermal hydraulic experiments.

10.1 Evaluation of Dissipation and Diffusion in the UTATF

Turbulent dissipation and diffusion are difficult to quantify from visual data such as the University of Tennessee Air Test Facility (UTATF) flow visualization experiments. Indeed such experiments are inherently more qualitative than quantitative. In order to obtain more quantitative observations of the turbulent dissipation in the recirculation zones, the UTATF was modified so that one of the two inlet streams could be heated to roughly 45 °C while the other inlet remained at room temperature. A high-speed thermocouple (1 mil in diameter) with a

response time on the order of milliseconds was used to measure variations in the local temperature at various positions in the UTATF flow field. All of the measurements were recorded at the center plane parallel to the test section bottom and at a distance of 3 inches downstream of the leading edge of the flow baffle. Measurements were recorded at the test section centerline, as well as 2 inches, 4 inches, 6 inches and 7.5 inches from the test section centerline, with the last measurement corresponding to the surface of the flow baffle. An additional data point was acquired in a closed container so that the contribution of electrical noise to the other signals could be evaluated. The recorded voltage signals from the thermocouple are shown in Figures 10.1 through 10.6. Clearly the thermocouple readings show that there is a great deal of oscillation in local temperature at each position.

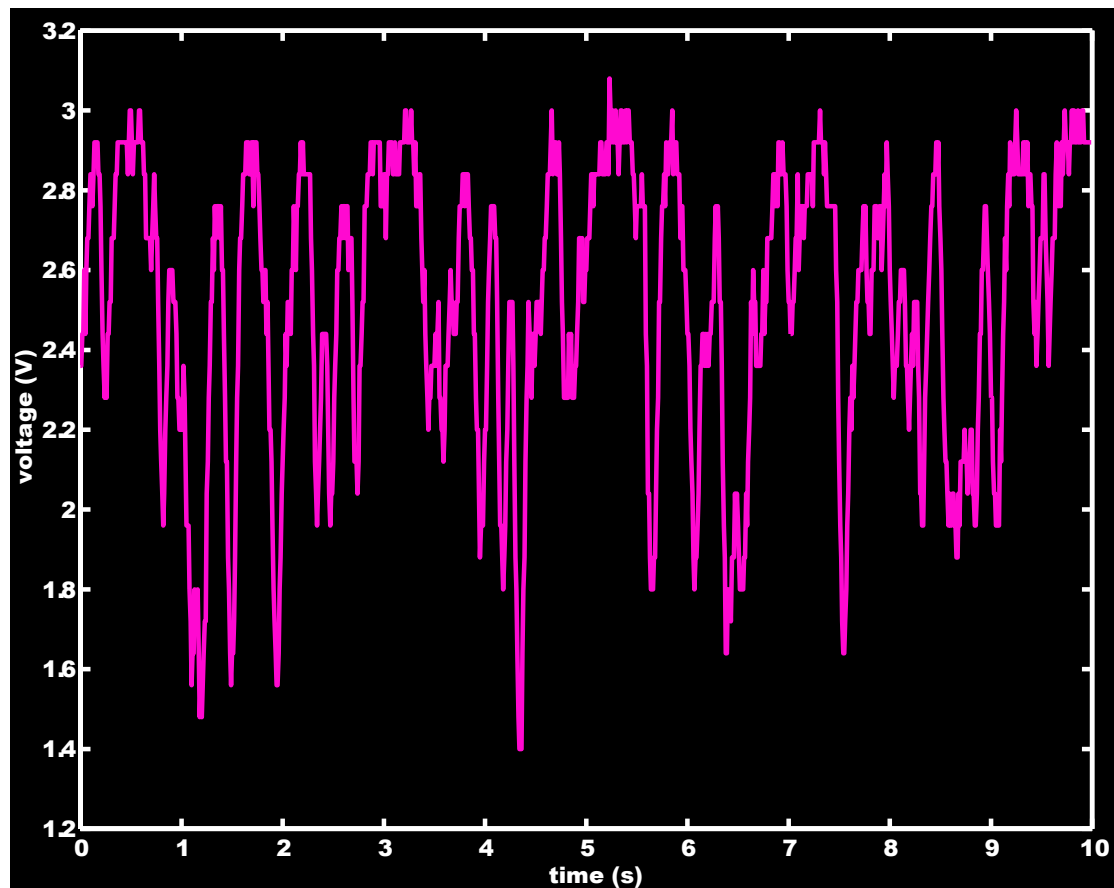


Figure 10.1 – Voltage signal from high-speed thermocouple located at the test section centerline.

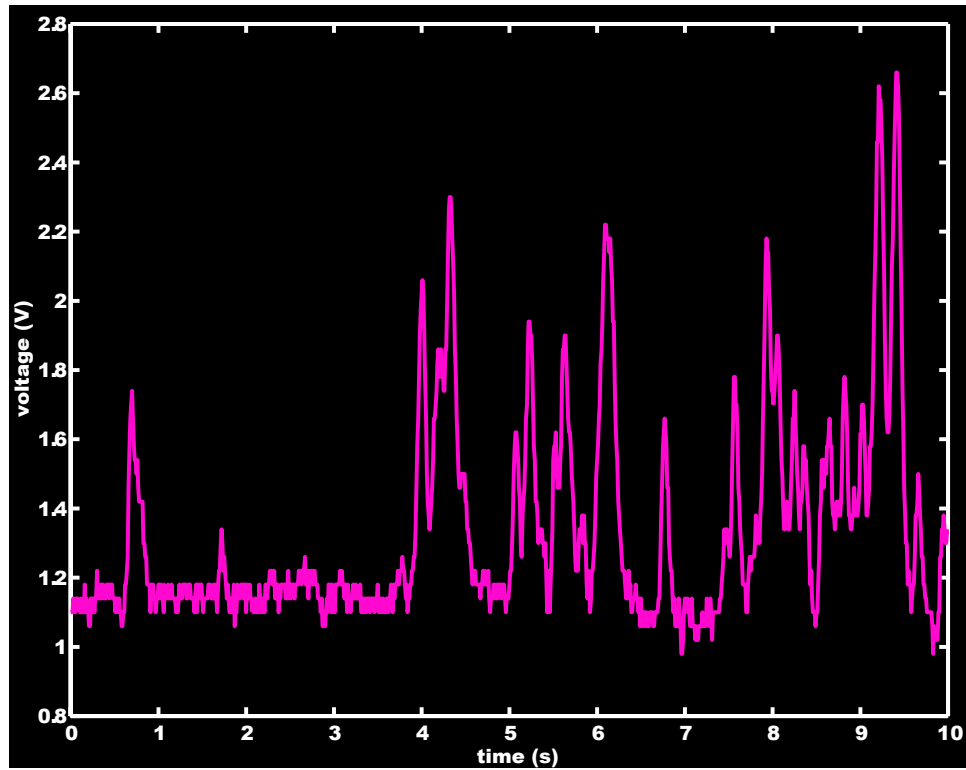


Figure 10.2 – Voltage signal from high-speed thermocouple located 2 inches from the test section centerline.

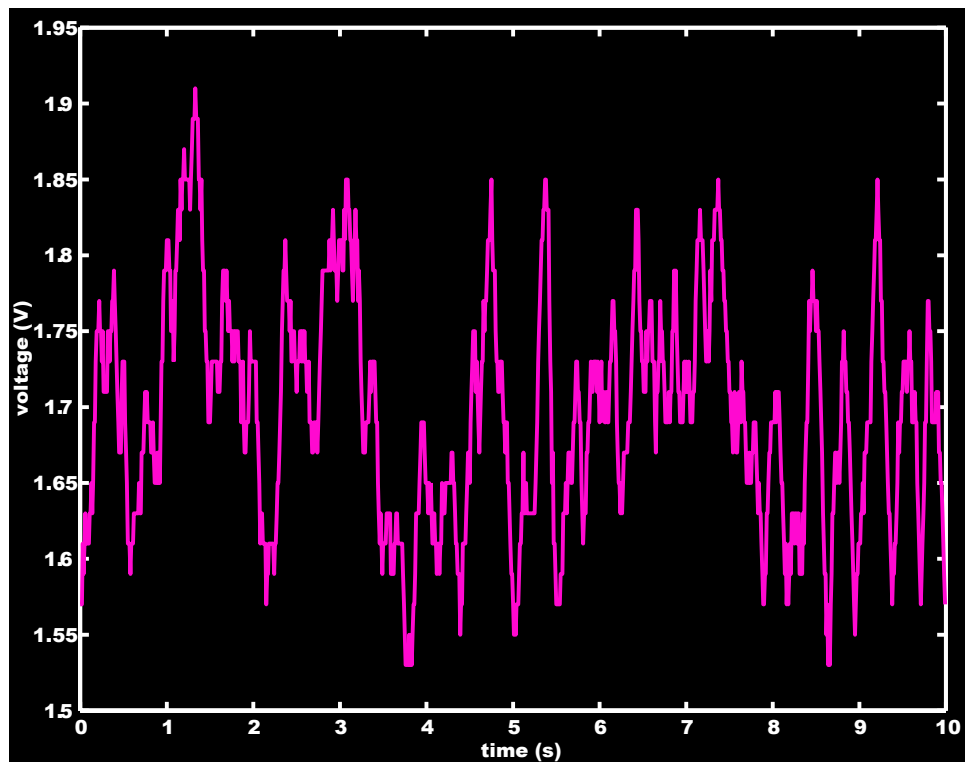


Figure 10.3 – Voltage signal from high-speed thermocouple located 4 inches from the test section centerline.

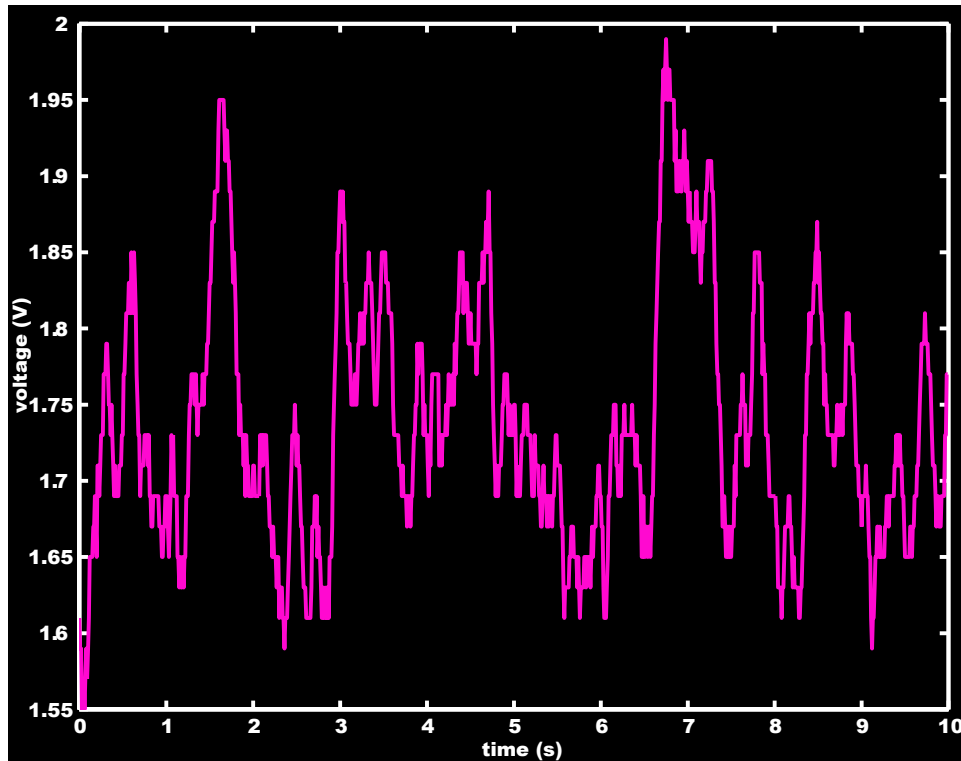


Figure 10.4 – Voltage signal from high-speed thermocouple located 6 inches from the test section centerline.

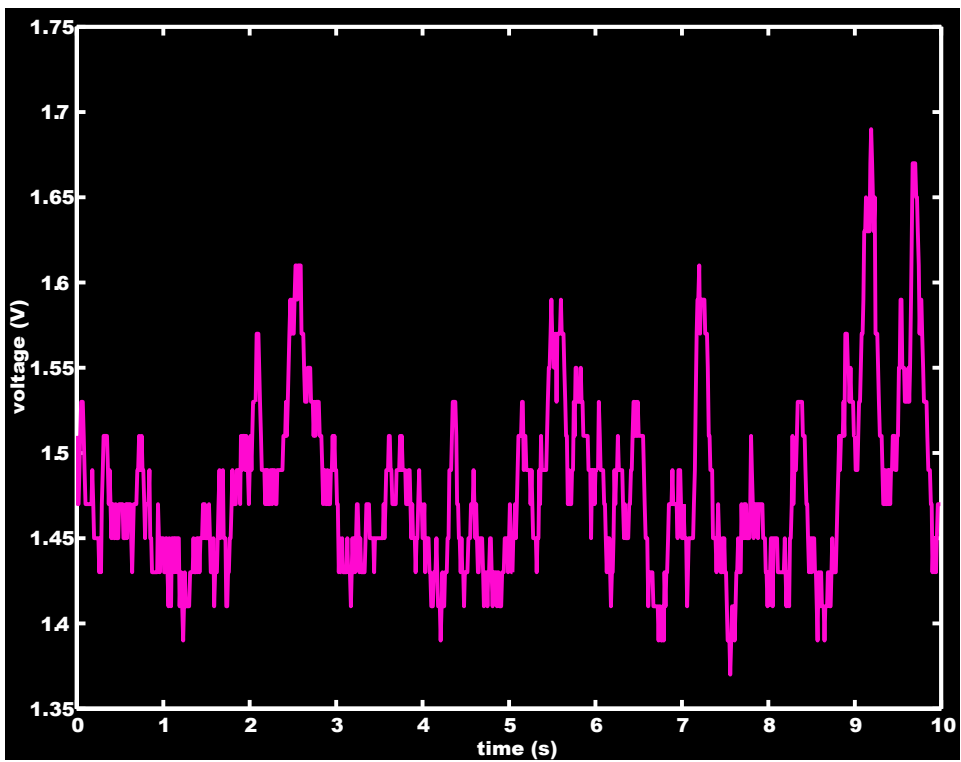


Figure 10.5 – Voltage signal from high-speed thermocouple located 7.5 inches from the test section centerline.

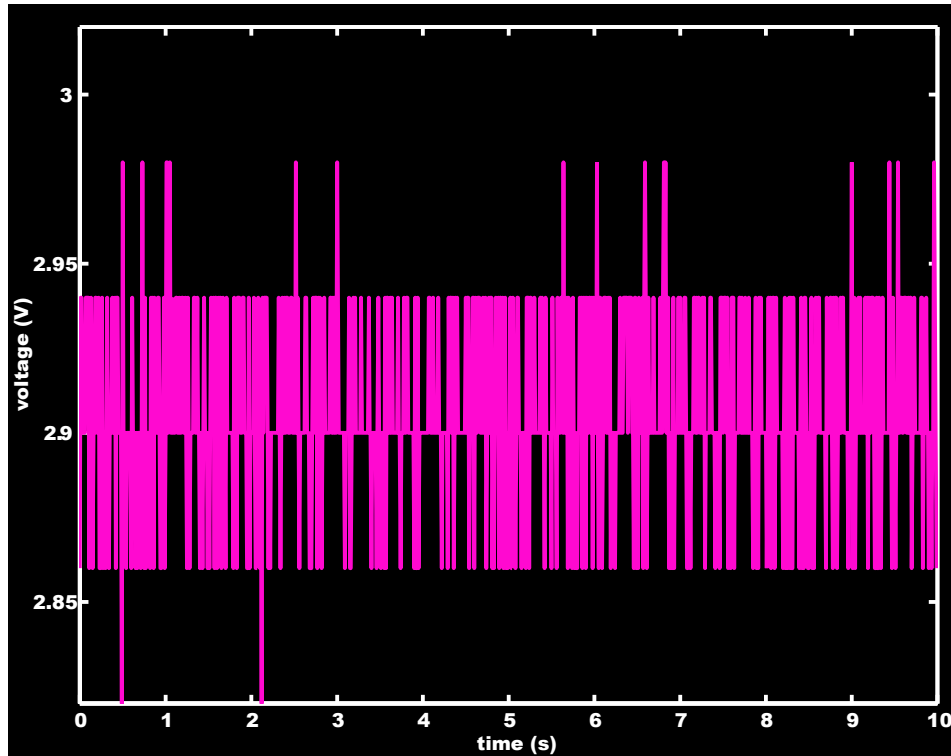


Figure 10.6 – Voltage signal from the control thermocouple located in a closed container. This signal is representative of the electromagnetic noise present in the other signals.

These signals can be used to determine how much of the oscillation results from the random turbulence in the flow field and how much results from larger periodic turbulence structures such as the primary recirculation zones. If the periodic flow behaviors dominate, the mixing patterns should exhibit periodic tendencies. The short-term Fourier transform provides a simple means of looking for evidence of periodic behavior in a signal. The short-term Fourier transform is calculated by dividing the signal into several overlapping windows and then performing a fast Fourier transform on the data contained in each window. The results of these evaluations are shown in graphical form in Figure 10.7 through Figure 10.12. In these representations, dark red indicates maximum frequency gains and blue indicates minimum frequency gains. While the short term Fourier transforms shown in Figures 10.7 and 10.8 may indicate some periodic behavior with frequencies on the order of 10 Hz, none of the transforms shown are remarkably different from the random noise seen in the transform shown in Figure 10.12. This would appear to indicate that the oscillations in the local temperature are the result of random turbulent mixing rather than periodic structures in the flow field. However the FFT method may fail to recognize very low frequency behaviors.

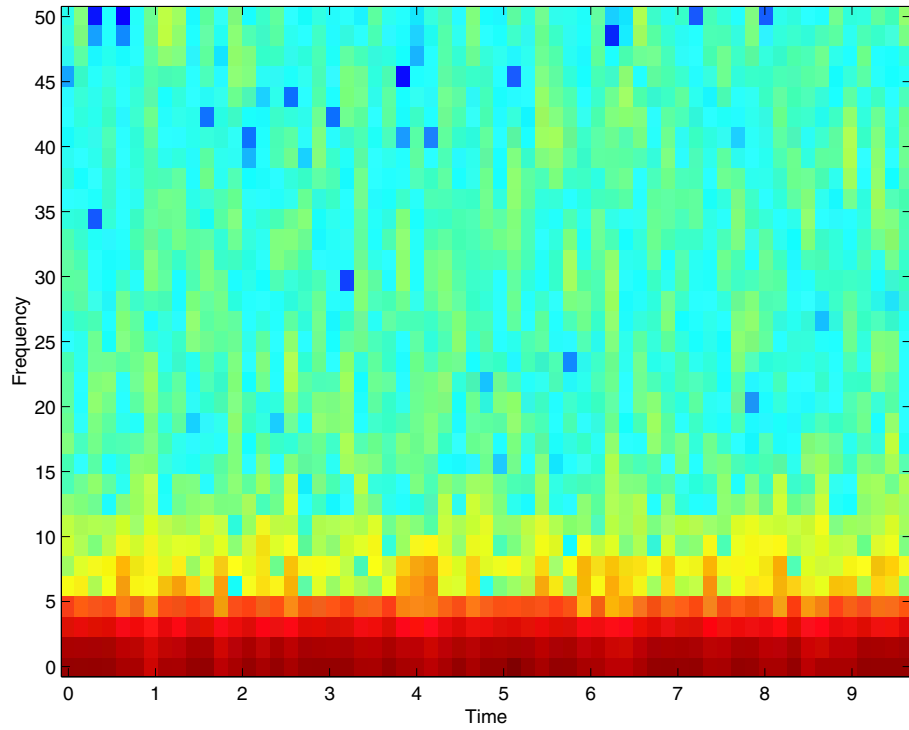


Figure 10.7 – Short term Fourier transform of voltage signal from high-speed thermocouple located at the test section centerline.

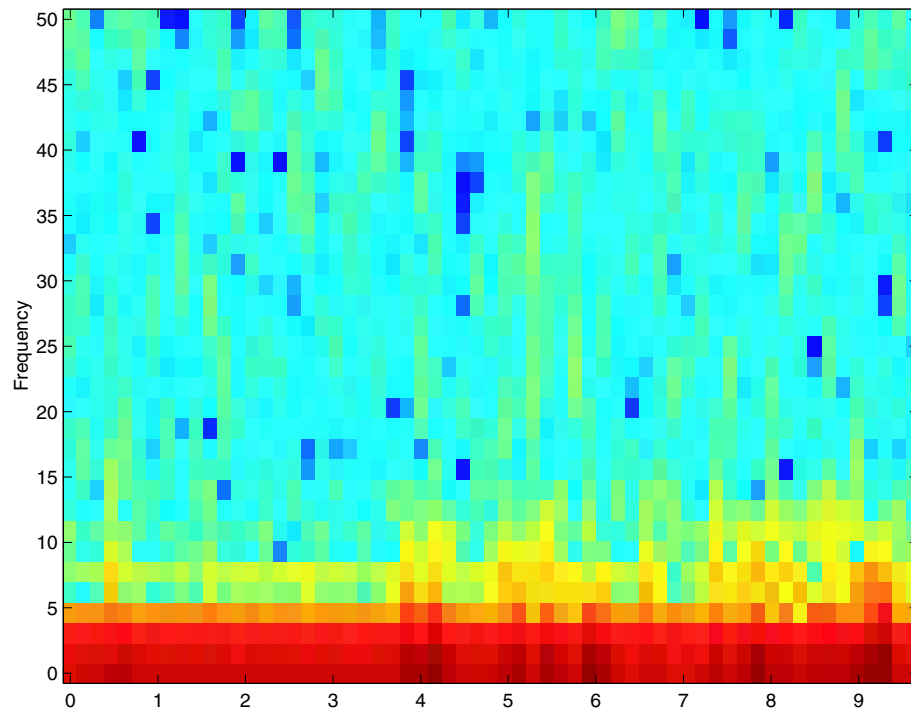


Figure 10.8 – Short term Fourier transform of voltage signal from high-speed thermocouple located 2 inches from the test section centerline.

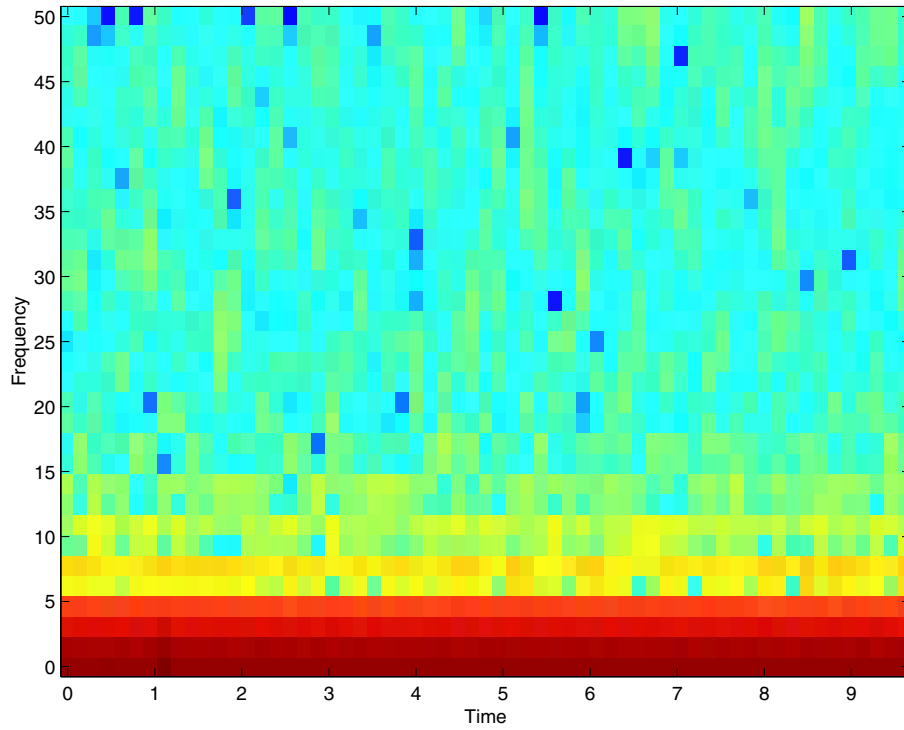


Figure 10.9 – Short term Fourier transform of voltage signal from high-speed thermocouple located 4 inches from the test section centerline.

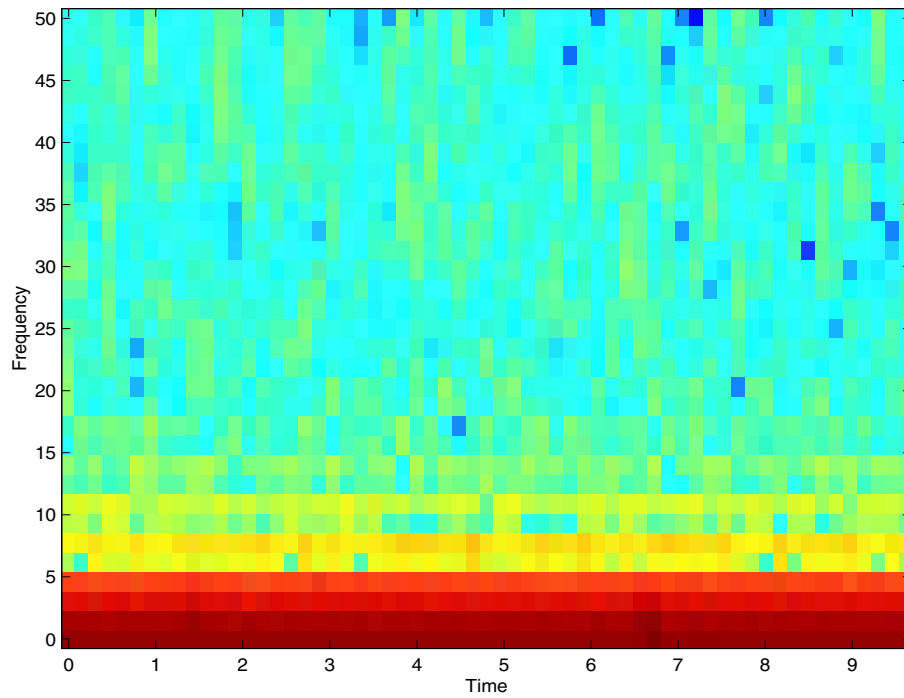


Figure 10.10 – Short term Fourier transform of voltage signal from high-speed thermocouple located 6 inches from the test section centerline.

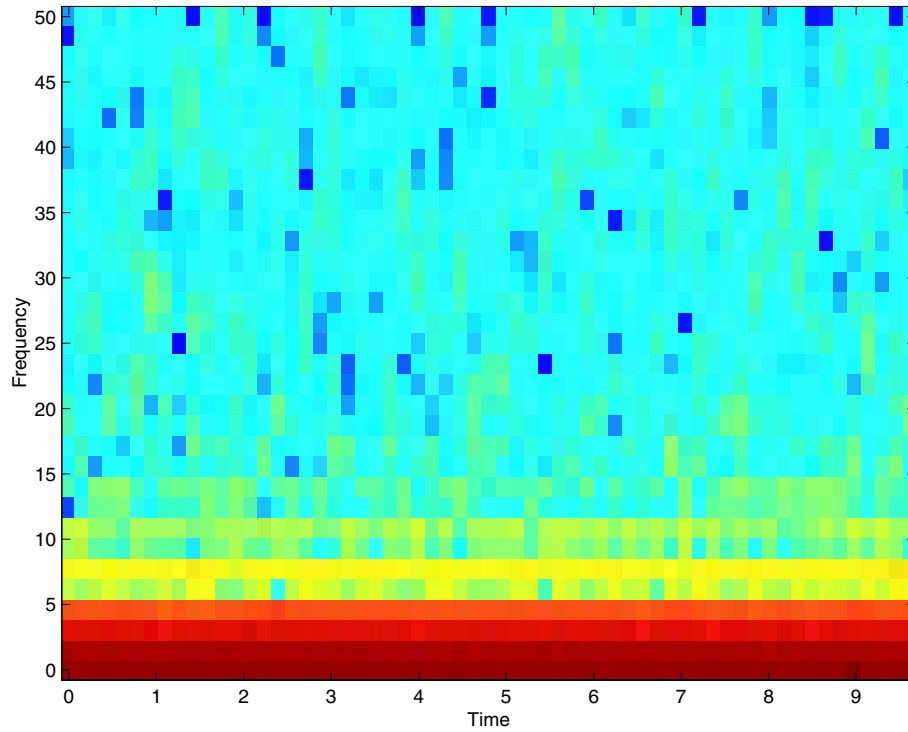


Figure 10.11 – Short term Fourier transform of voltage signal from high-speed thermocouple located 7.5 inches from the test section centerline.

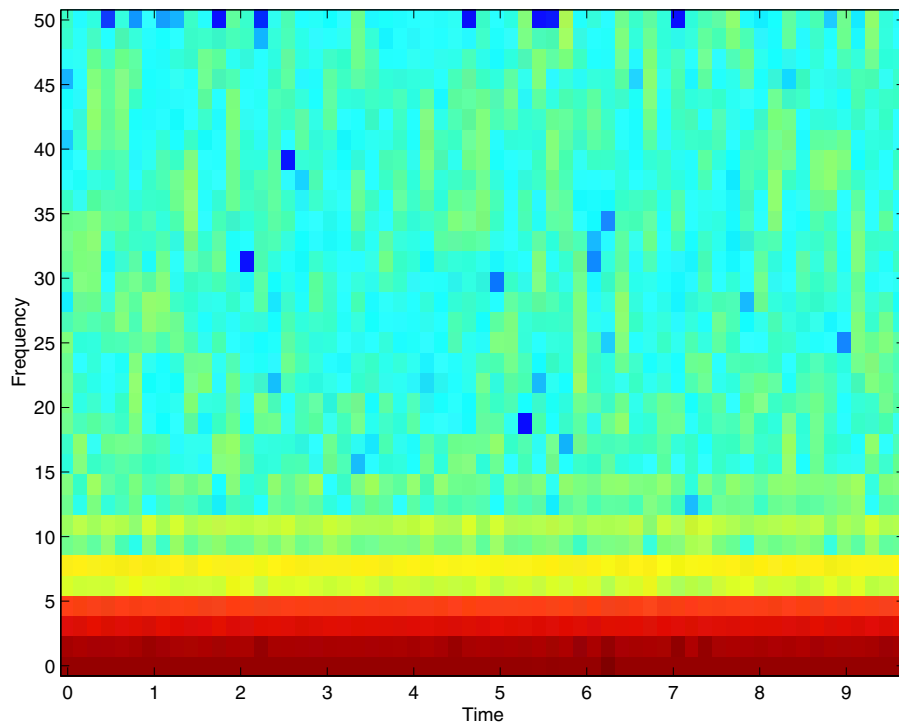


Figure 10.12 – Short term Fourier transform of voltage signal from the control thermocouple located in a closed container. This signal is representative of the electromagnetic noise present in the other signals.

The continuous 1-D wavelet transform provides a more robust measure of the periodicity of a signal than the short-term Fourier transform. When a wavelet transform is performed, the signal is decomposed in frequency space by fitting a series representation of a function to a segment of the signal. Each term in the series is based on some periodic function, called the wavelet package. Large coefficients will appear in a term of the fit function when a region of the signal exhibits periodicity of a form similar to the periodic function that appears in that term. The thermocouple signals from the UTATF experiments were decomposed using the Morlet Wavelet shown in Figure 10.13. Figures 10.14 through 10.19 show graphical representations of the continuous 1-D wavelet decomposition, where red indicates large coefficients and dark blue indicates small coefficients. When ridges of large coefficients appear in the wavelet transform, periodic tendencies are likely present in the original signal. Clearly, the wavelet transforms indicate the presence of periodic tendencies in the voltage signals from the thermocouples near the centerline of the test section (Figure 10.14 and Figure 10.15). However, no such tendencies are seen in the voltage signals from the thermocouples in the recirculation zone region or the control thermocouple. Thus, the fluctuations in the local temperature of the fluid in the recirculation zone regions of the UTATF test section are likely the result of random turbulent mixing in those regions.

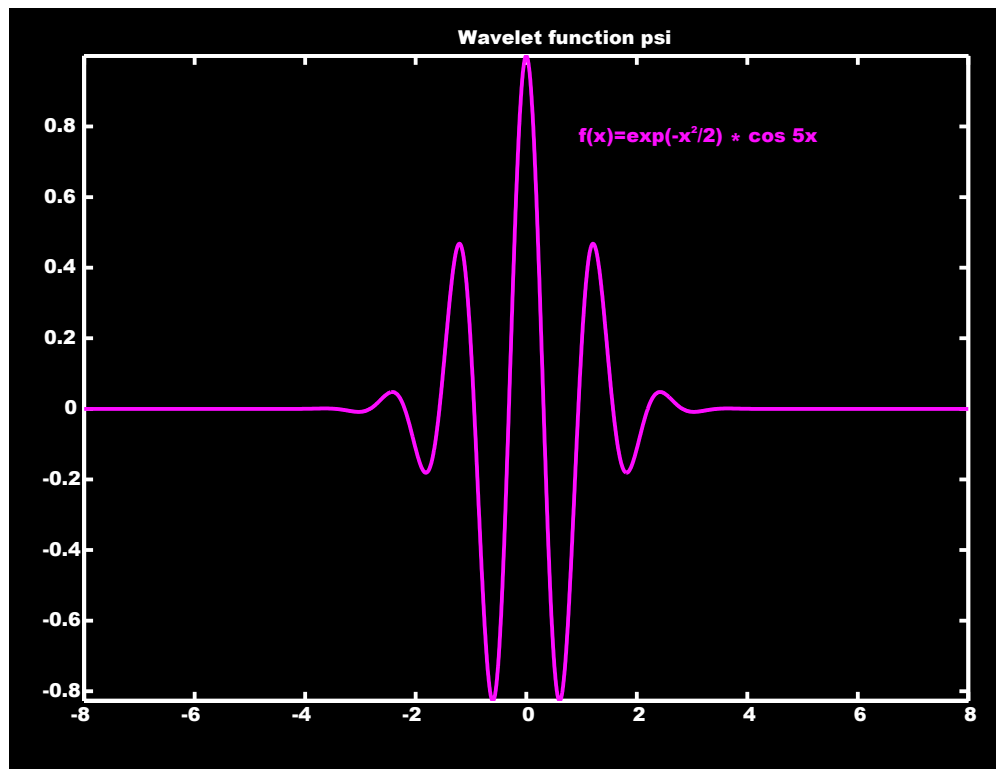


Figure 10.13 – Morlet wavelet used in the evaluation of voltage signals from the high-speed thermocouple used in the UTATF diffusion studies.

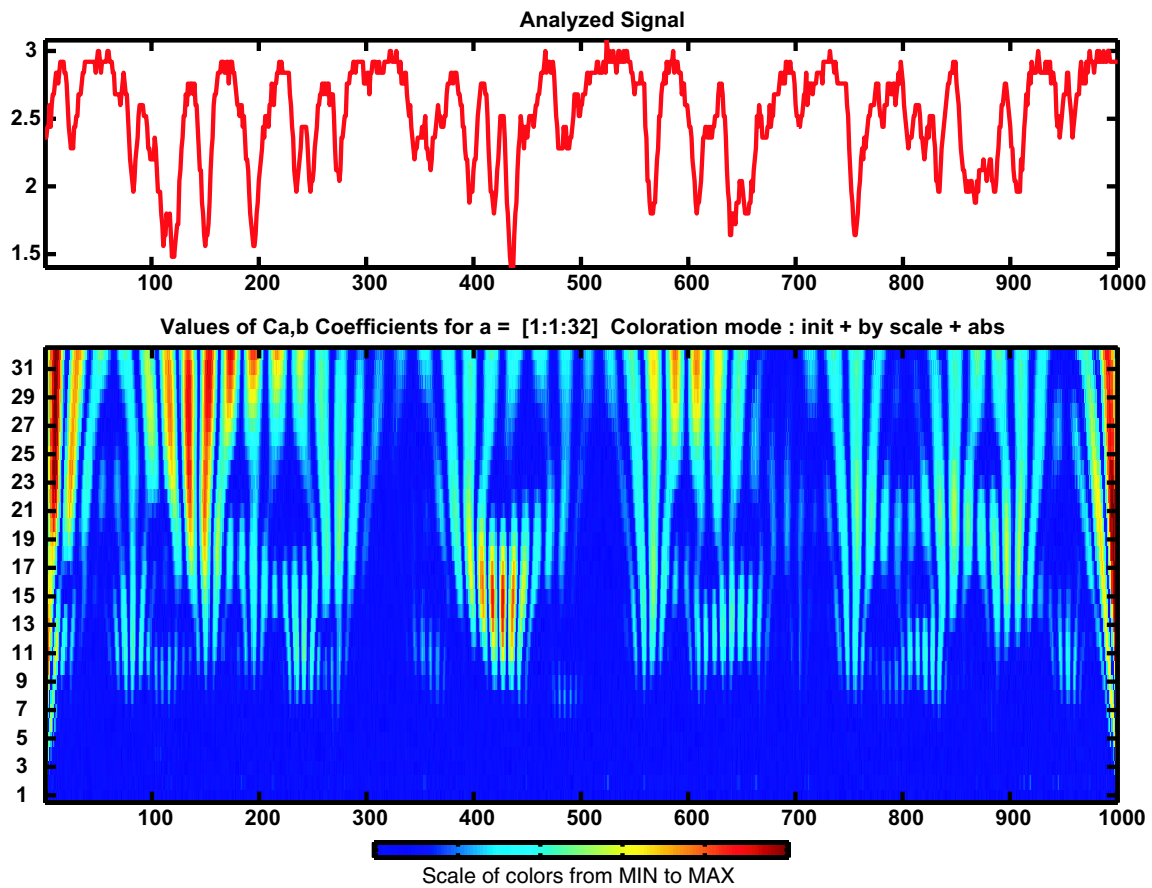


Figure 10.14 – Continuous 1-D wavelet transform of voltage signal from high-speed thermocouple located at the test section centerline.

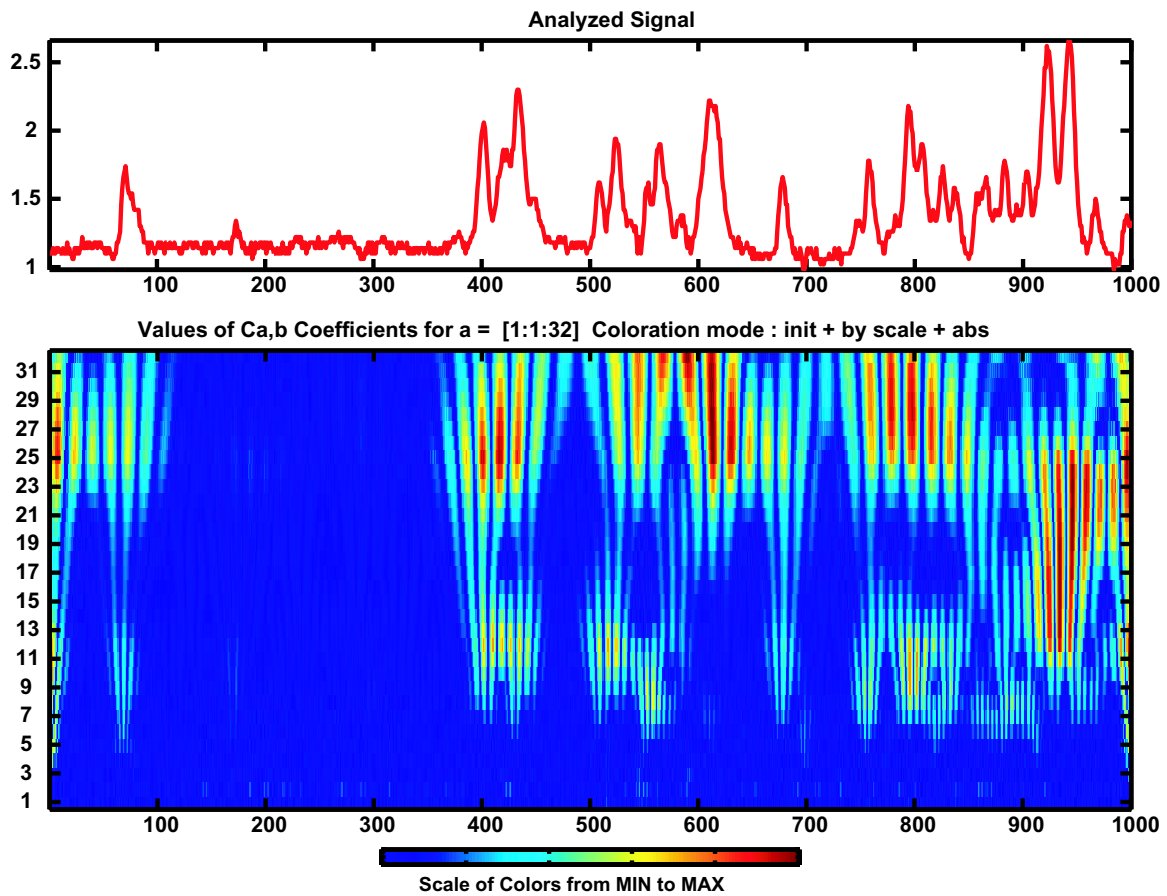


Figure 10.15 – Continuous 1-D wavelet transform of voltage signal from high-speed thermocouple located 2 inches from the test section centerline.

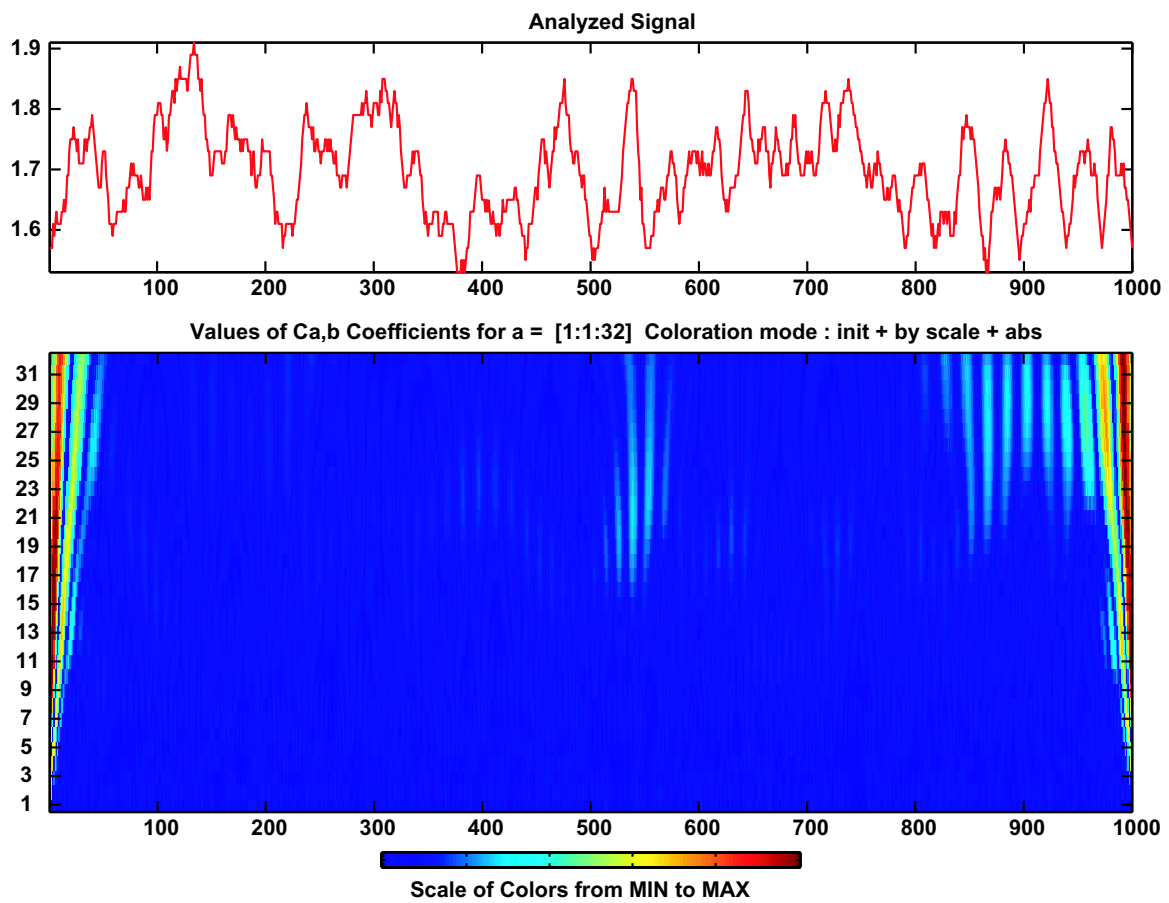


Figure 10.16 – Continuous 1-D wavelet transform of voltage signal from high-speed thermocouple located 4 inches from the test section centerline.

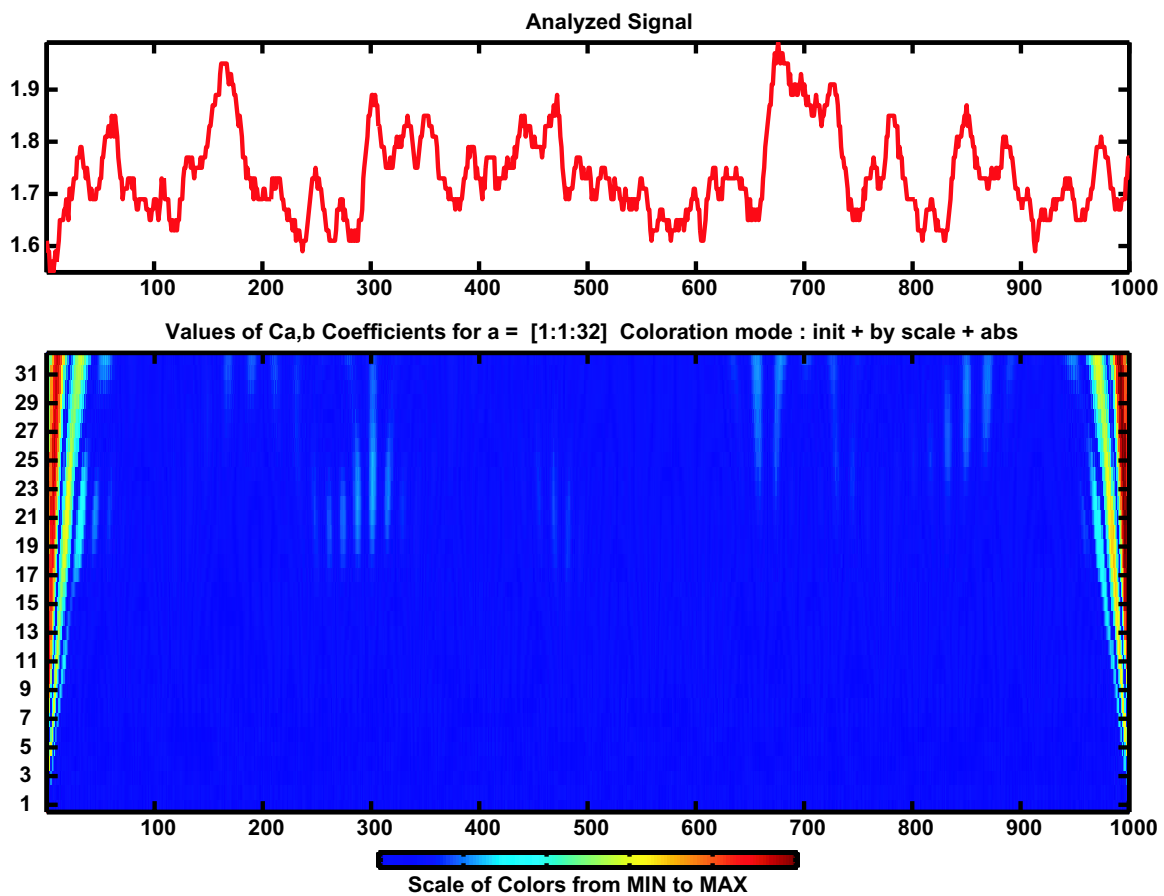


Figure 10.17 – Continuous 1-D wavelet transform of voltage signal from high-speed thermocouple

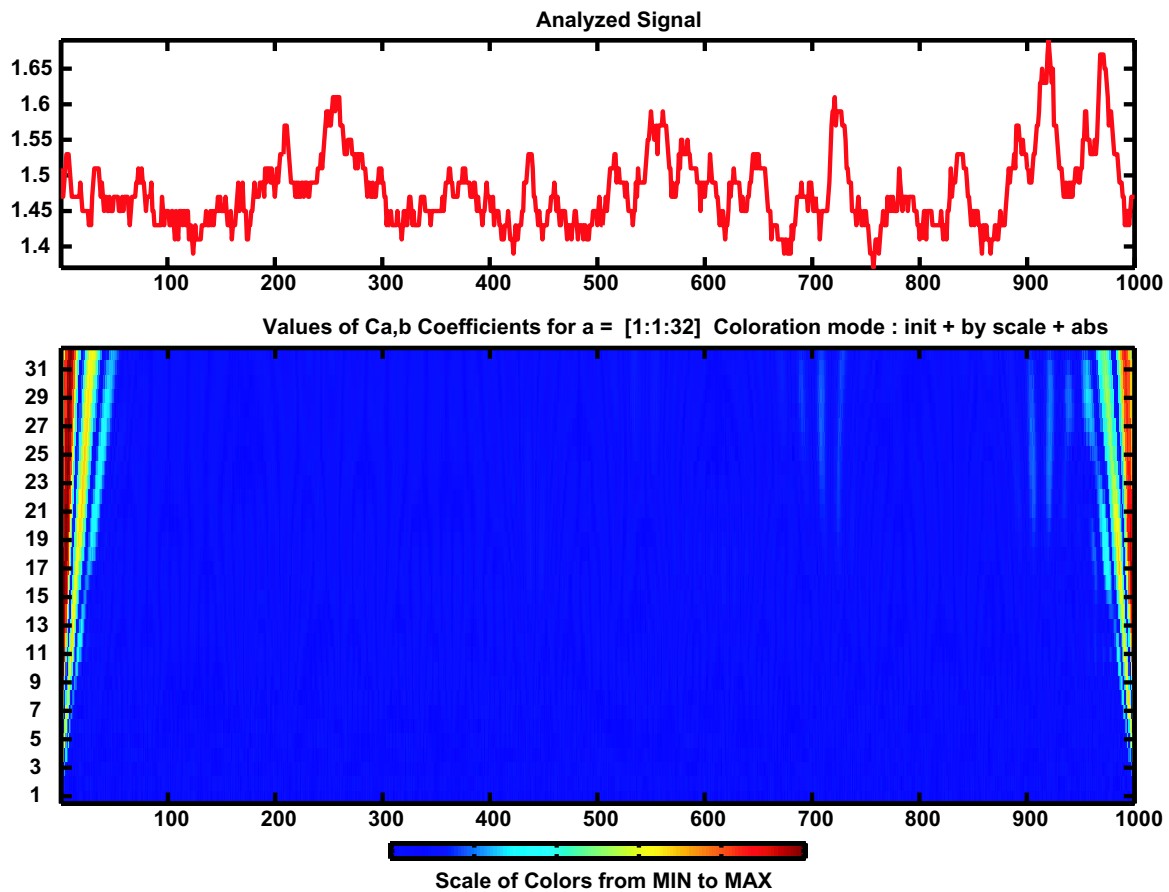


Figure 10.18 – Continuous 1-D wavelet transform of voltage signal from high-speed thermocouple located 7.5 inches from the test section centerline.

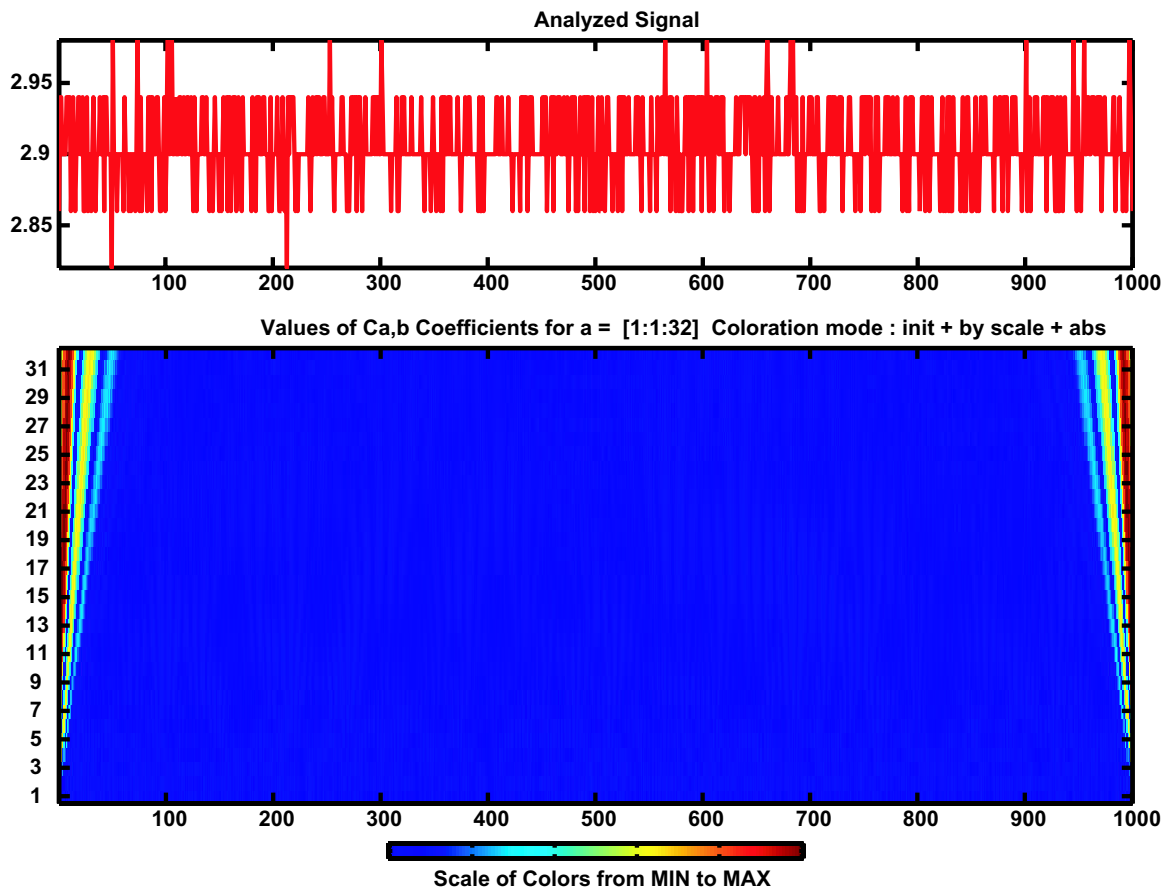


Figure 10.19 -- Continuous 1-D wavelet transform of voltage signal from the control thermocouple located in a closed container. The signal is representative of the electromagnetic noise present in other signals.

10.2 Evaluation of Dissipation and Diffusion in the WTHL

Studies in the UTATF indicate that the flow field in the recirculation zones is highly diffusive as a result of high turbulence intensity in those regions. While the UTATF provides a reasonable simulation of the SNS mercury target for identification of phenomena it is not ideally scaled for qualitative evaluations. In order to get a more accurate picture of the dissipation and diffusion in the recirculation zone regions, additional qualitative and quantitative experiments were performed in the WTHL, which is well scaled for the evaluation of the turbulent characteristics of the SNS flow field.

Visual Evaluation of Dissipation and Diffusion in the Recirculation Zone Regions

The flow visualization studies in the WTHL discussed in previous chapters were used to identify phenomena that occur in the flow field of the SNS mercury target. In these studies, one inlet flow is saturated with dye or bubbles, and mixing between the two inlet flows is observed. These studies are limited identification of diffusive patterns rather than quantification of diffusivity because the dye is not completely contained within individual flow field structures. By injecting the dye directly into the primary recirculation zones that develop along the leading edge of each flow baffle, the residence time of fluid in those regions can be observed directly.

A dye injection experiment for the evaluation of fluid residence times in the primary recirculation zones is shown in Figure 10.20. In these studies dye is injected for a period of approximately one second and the time required for the dye to be completely removed from the recirculation zone is observed. These studies indicate that the maximum residence time of a fluid element in the recirculation zone could be as long as a

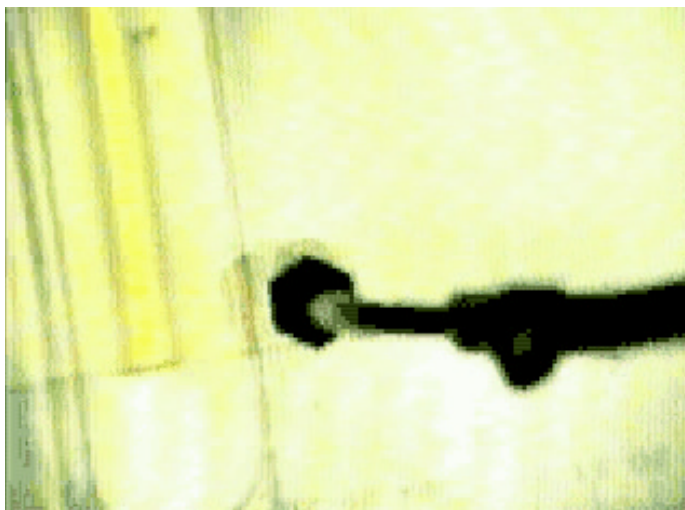


Figure 10.20 – Flow visualization experiment in the WTHL for evaluation of fluid residence times in the primary recirculation zones. To see video footage of the experiment, click on the image to activate the animation.

full second. Indicating that the individual fluid element could see as many as 60 proton pulses. While the WTHL is well scaled to reproduce the types of turbulence expected in the SNS target because the turbulence intensities are matched, the diffusivity of the flow field is not as well scaled because the turbulent kinetic energies are not as well matched. Thus, the residence time observed in these experiments should be considered as bounding the maximum possible residence time in the actual system.

Evaluation of Turbulent Fluctuations in the Velocity in the WTHL Test Section

While the LDV cannot map the velocity within the recirculation zones because of the limited field of vision through the acrylic windows, the UVP can measure velocities within those regions. Furthermore, the UVP can be used to make meaningful time dependent measurements that relate the variations in velocity in both time and space. Ultrasonic measurements were obtained at 9 positions in the WTHL Test Section. All measurements were made along the horizontal center plane of the test section as discussed in Chapter 8. Time-dependent UVP measurements in the WTHL at half-nominal flow rates can be seen in Figures 10.21 through 10.29. In these animations, data points that appear to indicate random full scale fluctuations are actually discrepancies that occur because the flow velocity is outside the measurement range of the UVP probe. These can be avoided by using a higher frequency probe, but spatial resolution is sacrificed because the width of the measurement area increases with the range of the probe. The measurements in the region nearest the baffle wall indicate that the maximum fluctuation in the velocity in that region is approximately 0.5 m/s. These fluctuations in the velocity cause oscillations in both the dissipation of fluid in the primary recirculation zone and the local heat transfer coefficient between the baffle wall and the bulk fluid flow.

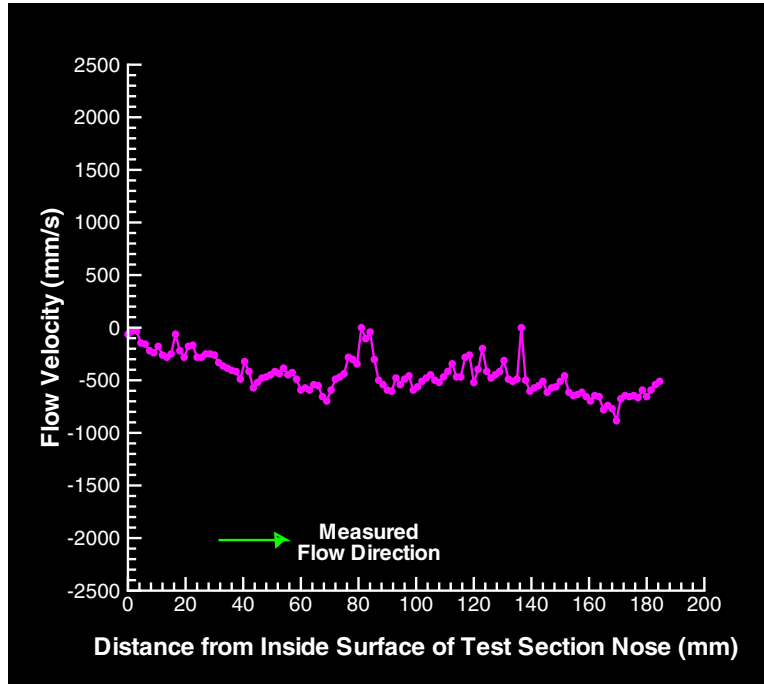


Figure 10.21 – Time-dependent velocity away from the test section nose at a distance of 4 inches to the left of the centerline of the exit channel. To activate the animation illustrating the time-dependent behavior, click on the image shown.

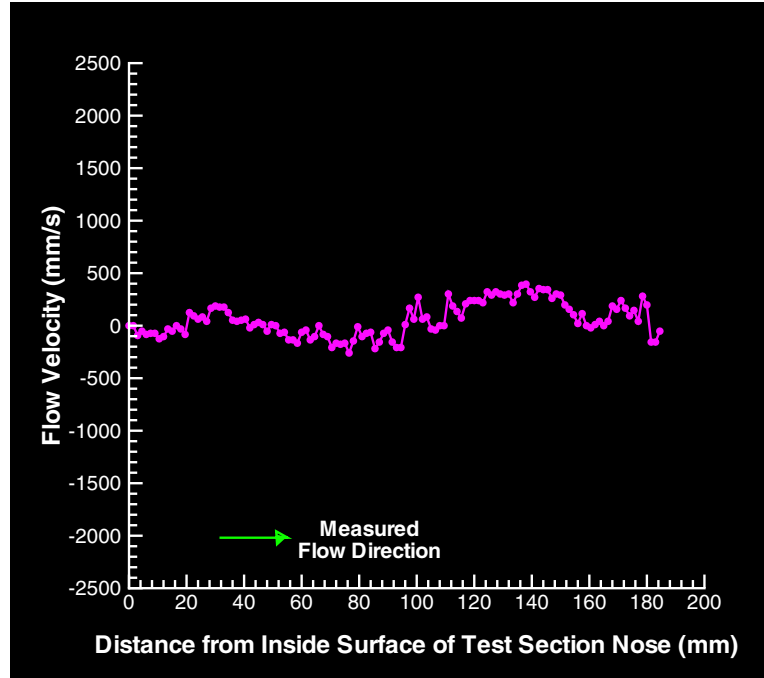


Figure 10.22 – Time-dependent velocity away from the test section nose at a distance of 3 inches to the right of the centerline of the exit channel. To activate the animation illustrating the time-dependent behavior, click on the image .

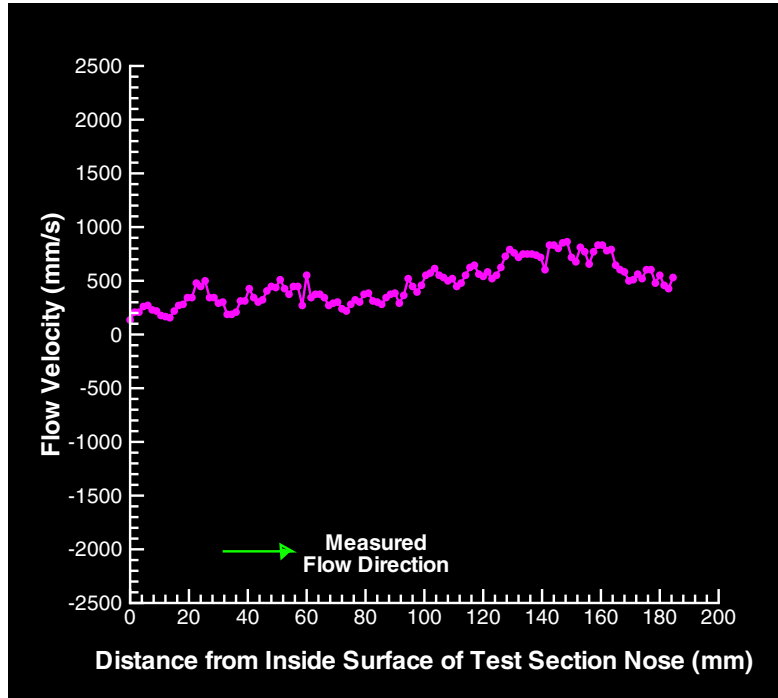


Figure 10.23 – Time-dependent velocity away from the test section nose at a distance of 2 inches to the right of the centerline of the exit channel. To activate the animation illustrating the time-dependent behavior, click on the image shown.

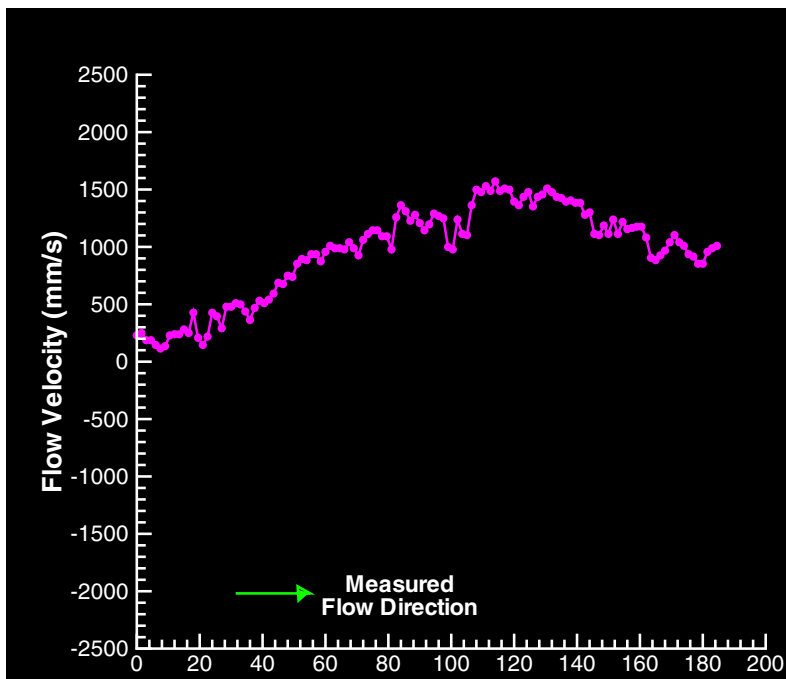


Figure 10.24 – Time-dependent velocity away from the test section nose at a distance of 1 inch to the right of the centerline of the exit channel. To activate the animation illustrating the time-dependent behavior, click on the image shown.

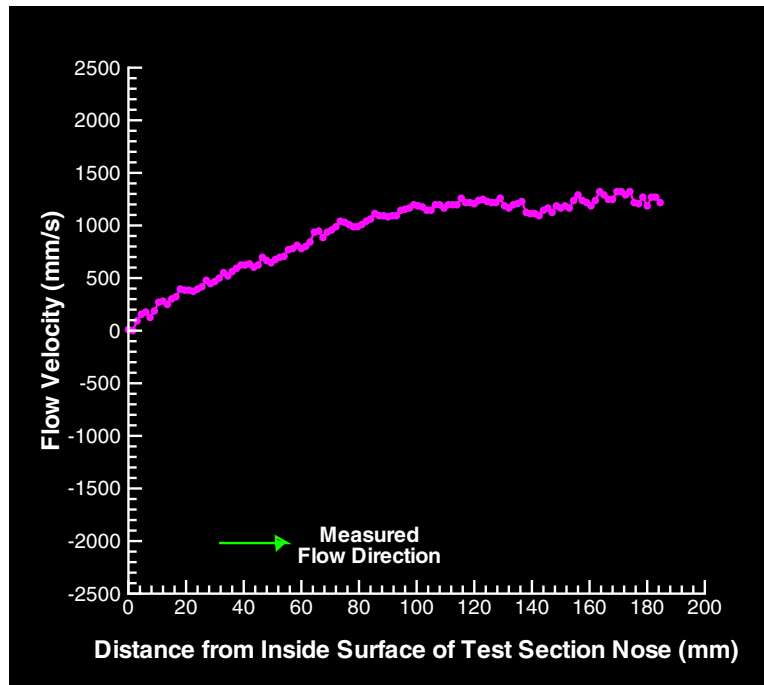


Figure 10.25 – Time-dependent velocity away from the test section nose at the centerline of the exit channel. To activate the animation illustrating the time-dependent behavior, click on the image shown.

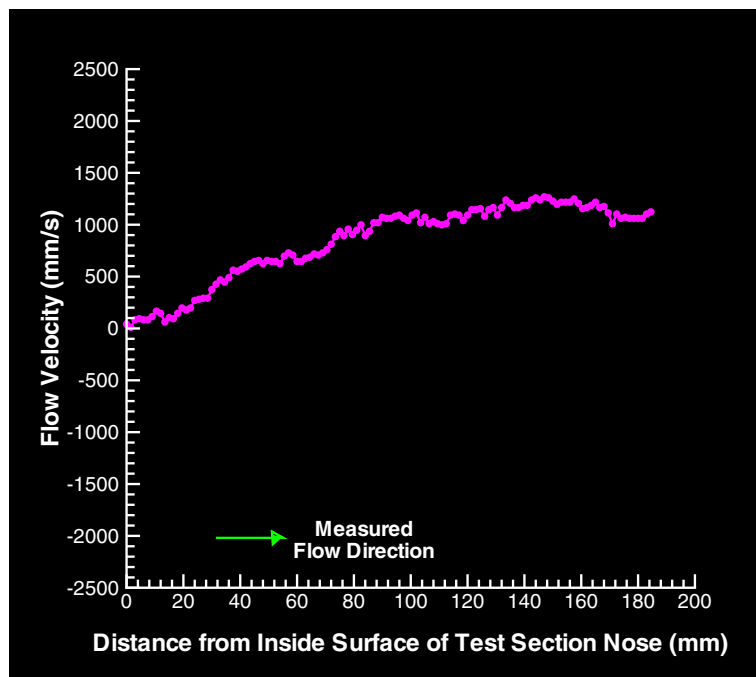


Figure 10.26 – Time-dependent velocity away from the test section nose at a distance of 1 inch to the left of the centerline of the exit channel. To activate the animation illustrating the time-dependent behavior, click on the image shown.

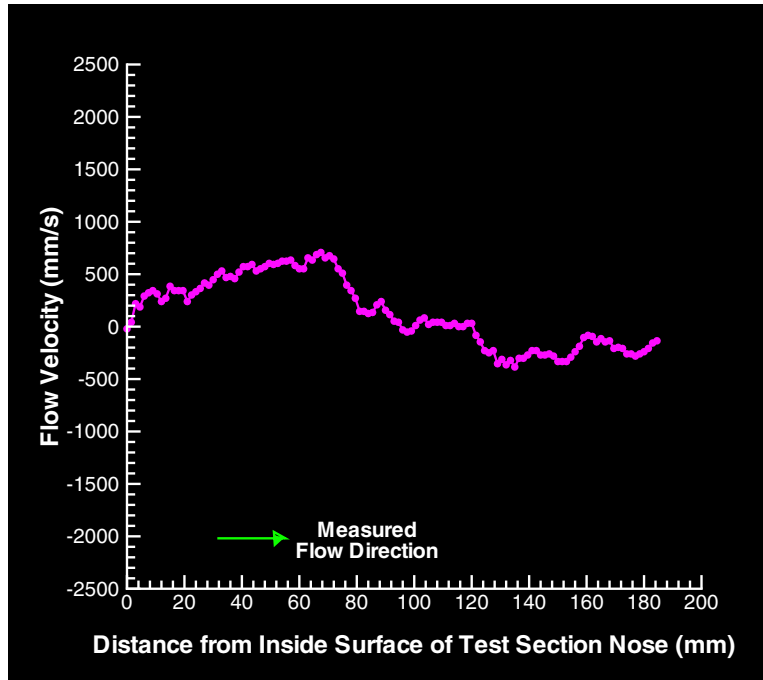


Figure 10.27 – Time-dependent velocity away from the test section nose at a distance of 2 inches to the left of the centerline of the exit channel. To activate the animation illustrating the time-dependent behavior, click on the image shown.

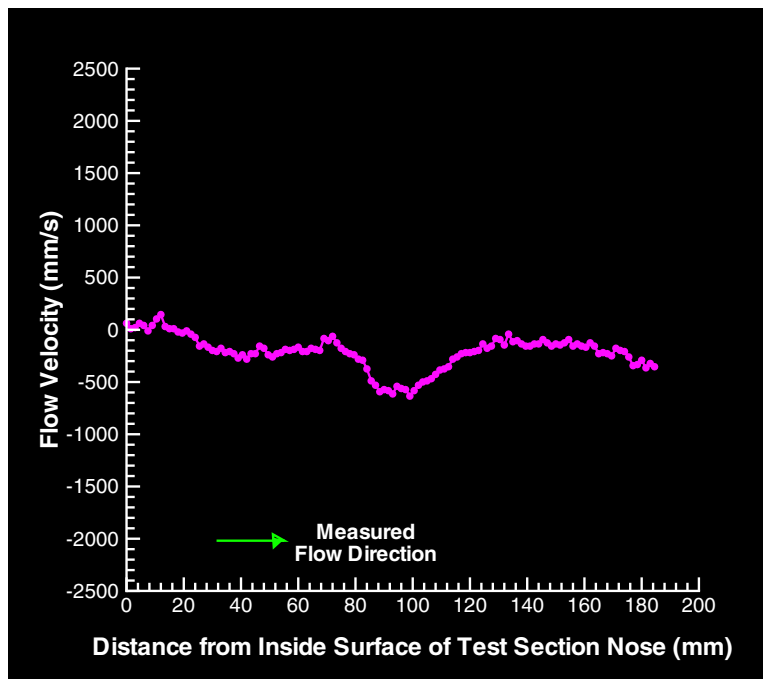


Figure 10.28 – Time-dependent velocity away from the test section nose at a distance of 3 inches to the left of the centerline of the exit channel. To activate the animation illustrating the time-dependent behavior, click on the image shown.

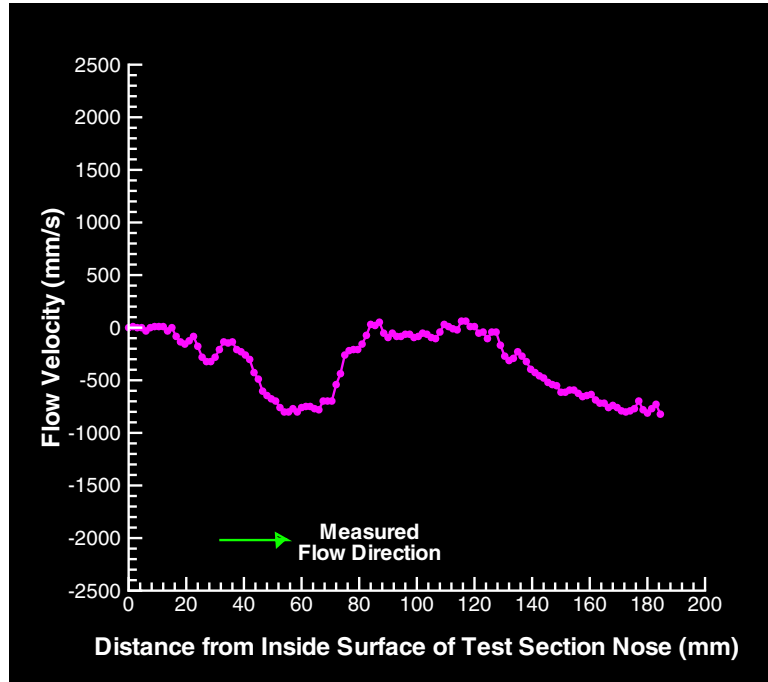


Figure 10.29 – Time-dependent velocity away from the test section nose at a distance of 4 inches to the left of the centerline of the exit channel. To activate the animation illustrating the time-dependent behavior, click on the image shown.

Chapter 11 Evaluation of Structural Temperature Distributions

11.1 Evaluation of Structural Temperature Distribution from CFD Models

Since the Computational Fluid Dynamics (CFD) model is the primary fluid mechanics design tool for the SNS mercury target, it is also the primary heat transfer design tool. The evaluation of the temperature of surface as a result of heated fluid moving past that surface requires knowledge of the velocity distribution in the fluid very near the wall. In this region, the turbulence models used in the remainder of the flow are no longer applicable, and the wall shear stress and the fluid viscosity determine the flow behavior. Therefore the velocity distribution may be determined by the equation known as the “law of the wall”, Equation 11.1.

Equation 11.1

$$\frac{\bar{U}}{u^*} = f\left(\frac{y u^*}{\nu}\right)$$

Through the use of a law of the wall approximation, the surface temperature of any portion of the mercury target that is in contact with the liquid mercury may be calculated based on the calculated turbulent flow behavior, the estimated near wall behavior and the power distribution in the heated flow field. The temperature distribution in the nominal mercury target is shown in Figure 11.1. Since the window region of the target is cooled by the secondary flow through the cooling channels, heat transfer from those surfaces to the bulk mercury flow is of little interest. The region of primary concern in the evaluation of structural temperatures based on this CFD model is the portion of each flow baffle that is exposed to fluid in the two primary recirculation zones. The temperature distribution on the surface of one of the two flow baffles also is shown in Figure 11.1. On a time average basis, the two flow baffles have identical temperature distributions. Based on these CFD simulations, the peak surface temperature occurring on the baffle surface is 180 °C.

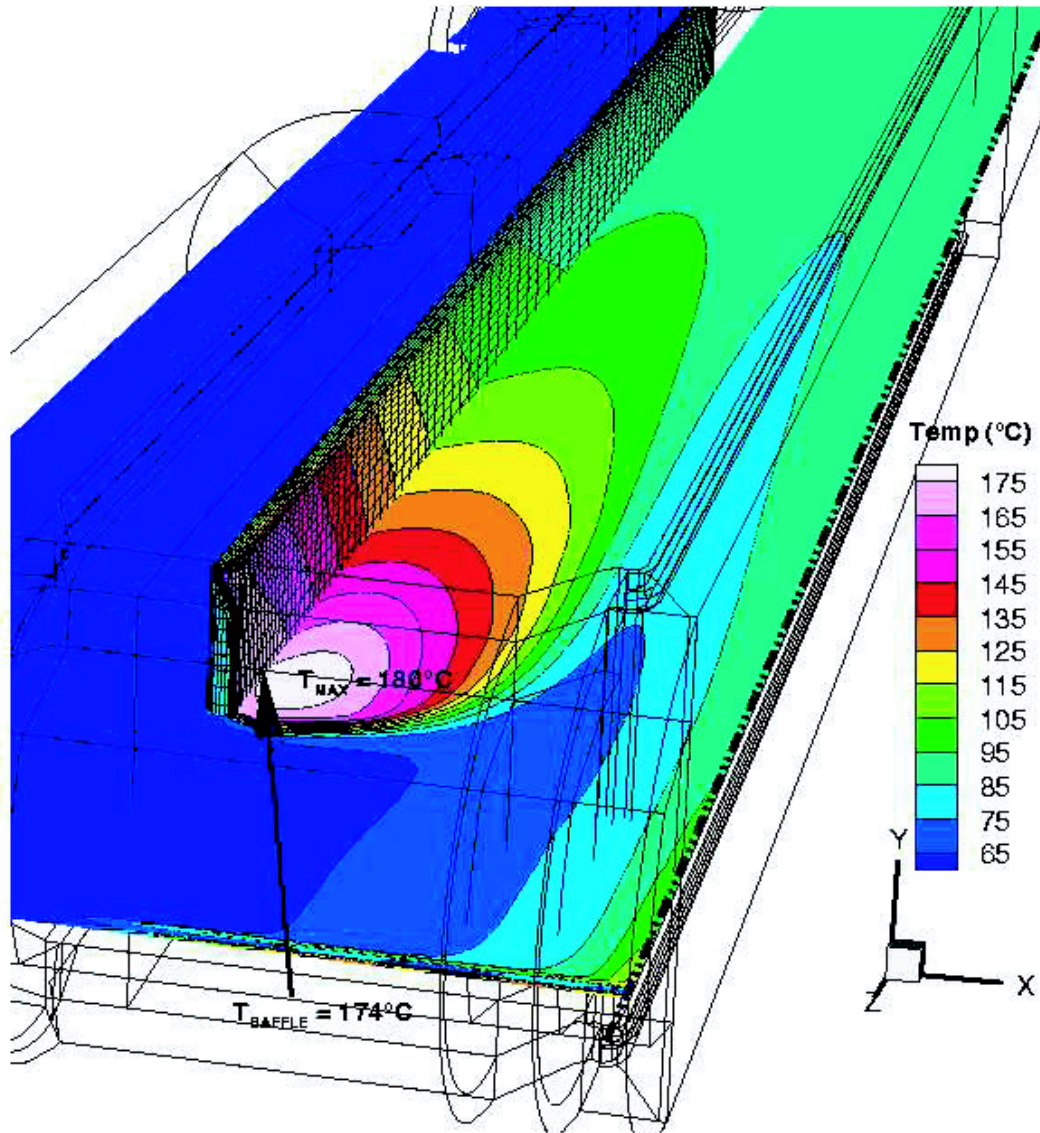


Figure 11.1 – Temperature distribution in the bulk mercury flow and on the surface of the flow baffle within the Spallation Neutron Source mercury target (Wendel, 2000 – Used with Permission).

11.2 Evaluation of Time-Dependent Fluctuation in the Baffle Wall Temperature

While the experimental studies discussed herein have shown that the CFD simulations provide a conservative model of the time-averaged behavior of the target flow field, it is impossible to estimate the time dependent fluctuations in the surface temperature of the flow baffles solely from the CFD results. For this reason, data from the thermal hydraulic experiments must be used to bound the limits of these fluctuations based on observed flow behaviors. Using an

engineering approach to the evaluation of the heat transfer from the fluid to the wall, the wall temperature can be evaluated using Equation 11.2.

Equation 11.2

$$T_w = T_{\text{Bulk}} - \frac{q''}{h}$$

The fluctuation of the wall surface temperature can be evaluated using Equation 11.3, which is a simplified perturbation form of Equation 11.2.

Equation 11.3

$$\Delta T_w = \Delta T_{\text{Bulk}} - \left(\frac{q''}{h + \Delta h} - \frac{q''}{h} \right)$$

The heat flux through the walls of the exit channel, q'' , can be approximated by assuming that the entire 2 MW deposited in the exit channel flow must exit the target through the surface area of the upper and lower walls of the exit channel plus the surface area of the two flow baffles. The resulting constant heat flux is approximately $5.56 \times 10^6 \text{ W/m}^2$. The remaining parameters, the variation in the heat transfer coefficient and the variation in the bulk fluid temperature, are a function of the observed fluid behavior.

Evaluation of Fluctuation in the Bulk Fluid Temperature

Local fluctuations in the bulk temperature of the liquid mercury occur throughout the SNS target as a result of the turbulent nature of the flow field and the time dependence of the heat addition. The bulk temperature fluctuations that directly affect the temperature of the baffle surface are those that occur in the primary recirculation zones. The fluid temperature fluctuations that occur as a result of the turbulent mixing of the flow field are relatively small in comparison to the magnitude of the average fluid temperature. However, in the recirculation zones an additional fluctuation in the fluid temperature results from the fluctuation in the residence time of a fluid element within the recirculation zone. As a result, some fluid elements may be exposed to the energy deposited by the proton beam longer than others may.

The energy balance for a fluid element in the recirculation region of the target is given by Equation 11.4.

Equation 11.4

$$q''' = \rho c_p \frac{\partial T}{\partial \tau}$$

This representation of the energy balance takes no credit for the heat transferred out of the fluid element as a result of turbulent diffusivity because it is the turbulent nature of the flow that results in the local variations in the residence time. With this in mind, the change in the temperature of the fluid as a result of the variation in residence time is given by Equation 11.5.

Equation 11.5

$$\Delta T = \frac{q''' \Delta \tau}{\rho c_p}$$

The average volumetric heat generation rate in the recirculation zone region is on the order of $4 \times 10^8 \text{ W/m}^3$. For this rate of heat generation, the variation in the bulk fluid temperature is shown as a function of the variation in residence time in Figure 11.2, where the properties of the liquid mercury are evaluated at 150°C . Based on direct observation from the flow visualization studies in the Water Thermal Hydraulic Loop (WTHL), the maximum possible residence time in the

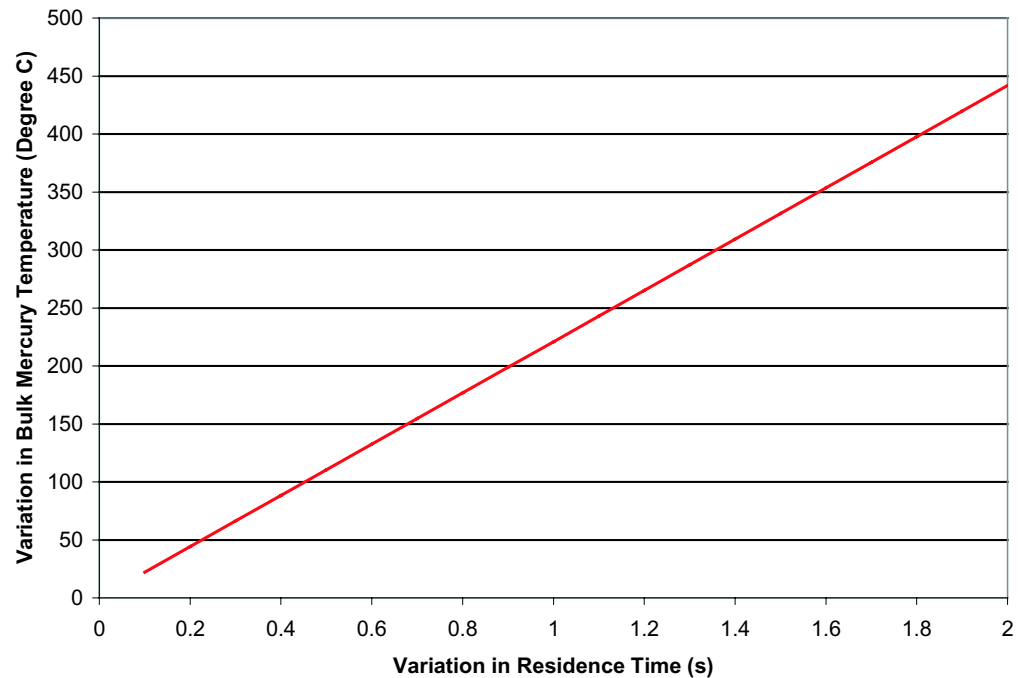


Figure 11.2 – Local variation in bulk fluid temperature as a function of variation in fluid residence time.

recirculation zones of the WTHL test section is roughly 0.5 seconds longer than the residence time of the surrounding fluid. Since the WTHL experiments are scaled to match the turbulent diffusivity of the prototypic mercury target, maximum possible residence times should be similar in both facilities. These observations indicate that the maximum possible variation in the bulk fluid temperature is approximately 110 °C.

Evaluation of Fluctuation in the Heat Transfer Coefficient

The inherent turbulent nature of the flow field also results in local fluctuations in the heat transfer coefficient between the bulk mercury flow and the surfaces of the target structure. Since the changes in the properties of liquid mercury are rather small for the range of temperature variations expected in the bulk mercury flow, the primary source of variation in the heat transfer coefficient is the variation in the flow velocity in the recirculation zone region. As in the previous evaluation, no credit can be taken for the turbulent diffusivity since its effect is being explicitly considered when the variation in heat transfer coefficient resulting from turbulent variations in the flow velocity is evaluated.

For fluids of small Prandtl number, such as liquid metals, the thermal boundary layer develops much more rapidly than the velocity boundary layer as a result of the high conductivity of the fluid. Therefore, it is reasonable to assume a uniform velocity distribution throughout the thermal boundary layer for most flow fields. Furthermore, on the scale of interest, the flow baffle may be considered as a flat plate in parallel flow, subject to the energy balance given by Equation 11.6.

Equation 11.6

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \alpha \frac{\partial^2 T}{\partial y^2}$$

Rewriting this equation in its non-dimensional form and solving using the assumption of a uniform velocity distribution yields the heat transfer correlation shown in Equation 11.7 (Incropera and Dewitt, 1990).

Equation 11.7

$$Nu_x = \frac{h x}{k} = 0.565 Pe_x^{1/2}$$

Using the definition of the Peclet number, Pe , the correlation can be evaluated to find the heat transfer coefficient of the form shown in Equation 11.8.

Equation 11.8

$$h_x = 0.565 \frac{k}{x} \text{Re}_x^{1/2} \text{Pr}^{1/2}$$

This equation can be used to determine the relationship between flow velocity and the transfer coefficient for the recirculation zone region as shown in #Equation 11.9. The model is tuned to the application by recognizing that the characteristic dimension of interest is the half-width of the primary recirculation zone, d_c .

Equation 11.9

$$h = 0.565 \frac{k}{d_c} \left(\frac{v d_c}{\mu} \right)^{1/2} \text{Pr}^{1/2}$$

Consequently, the variation in the heat transfer coefficient resulting from the variation in the flow velocity in the segment of the primary recirculation zone that is near the baffle wall can be evaluated using Equation 11.10, where it is assumed that the properties of the fluid remain constant.

Equation 11.10

$$\Delta h = 0.565 \frac{k}{d_c} \left(\frac{\Delta v d_c}{\mu} \right)^{1/2} \text{Pr}^{1/2}$$

The variation of the heat transfer coefficient in the recirculation zone region is shown as function of the variation in velocity in Figure 11.3.

Evaluation of Fluctuations in Baffle Wall Temperature resulting from Fluctuations in Bulk Fluid Temperature

The local variation of the bulk fluid temperature directly affects the temperature of the baffle surface, as is evident in Equation 11.3. If the variation in the heat transfer coefficient is assumed to be negligible, Equation 11.3 can be reduced to Equation 11.11.

Equation 11.11

$$\Delta T_w = \Delta T_{\text{Bulk}}$$

Using this equality, the variation in the surface temperature of the baffle can be evaluated as shown in Figure 11.4. Since the maximum expected variation in the bulk fluid temperature is approximately 110 °C, the maximum variation in the wall temperature resulting from variations in the bulk temperature is also 110 °C.

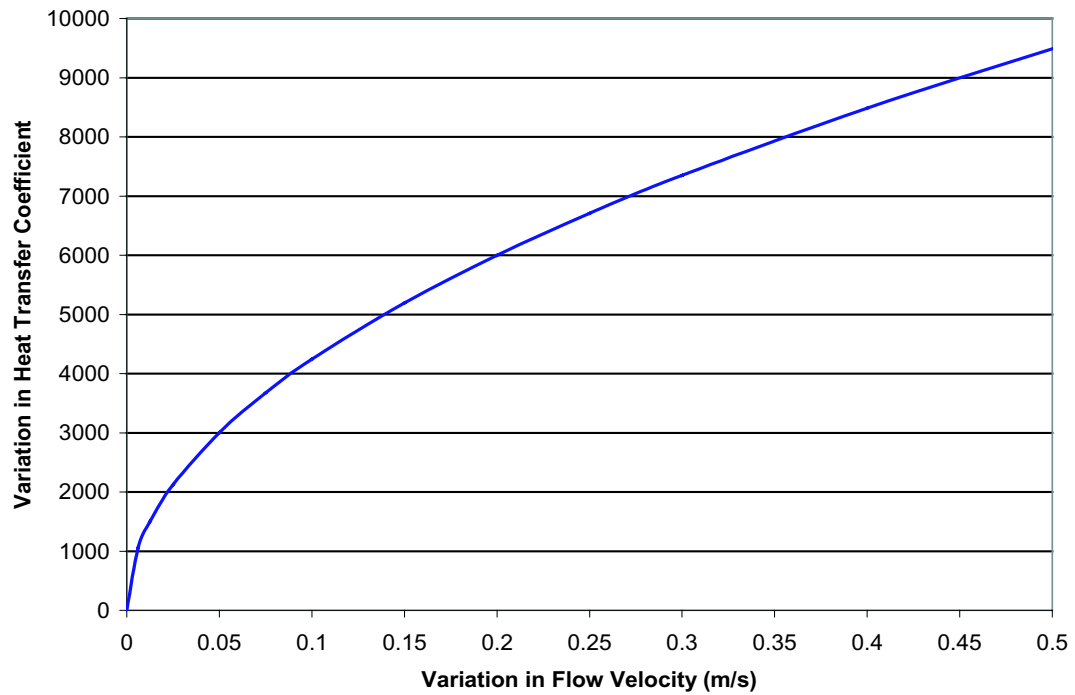


Figure 11.3 – Local variation of the heat transfer coefficient as a function of the magnitude of variation in the flow velocity.

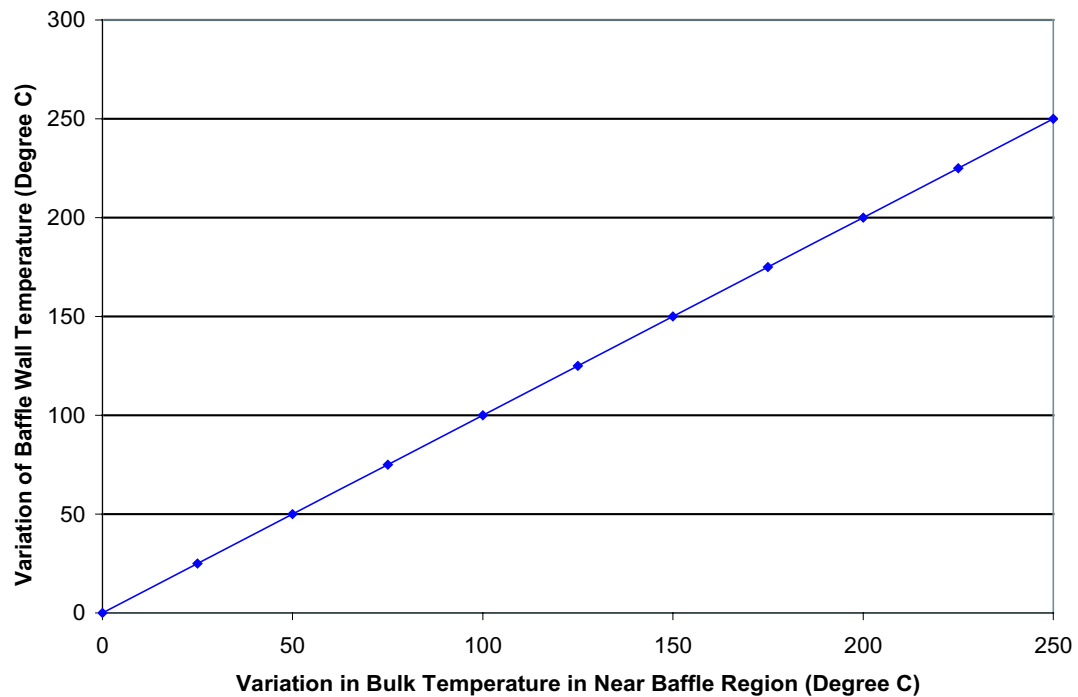


Figure 11.4 – Variation in the surface temperature of the flow baffle as a result of variation in the bulk fluid temperature.

Evaluation of Fluctuations in Baffle Wall Temperature resulting from Fluctuations in Flow Velocity

The local variation in the flow velocity also affects the surface temperature of the flow baffle through the variations in the heat transfer coefficient that result from the changes in the flow velocity. If the effects of the variation in the bulk fluid temperature are assumed to be constant, Equation 11.3 can be rewritten in the form shown in Equation 11.12.

Equation 11.12

$$\Delta T_w = \left(\frac{q''}{h + \Delta h} - \frac{q''}{h} \right)$$

Using this relation, the variation in the surface temperature of the flow baffle can be evaluated as a function of the fluctuation of the heat transfer coefficient, which has already been shown to be a function of the variation in the flow velocity. The variation in the surface temperature resulting from fluctuations in the flow velocity is shown in Figure 11.5. The maximum fluctuation in flow velocity in the region near the flow baffle is approximately 0.5 m/s, based on Ultrasonic Velocity Profiliometer measurements in that region. Consequently, the maximum fluctuation in the

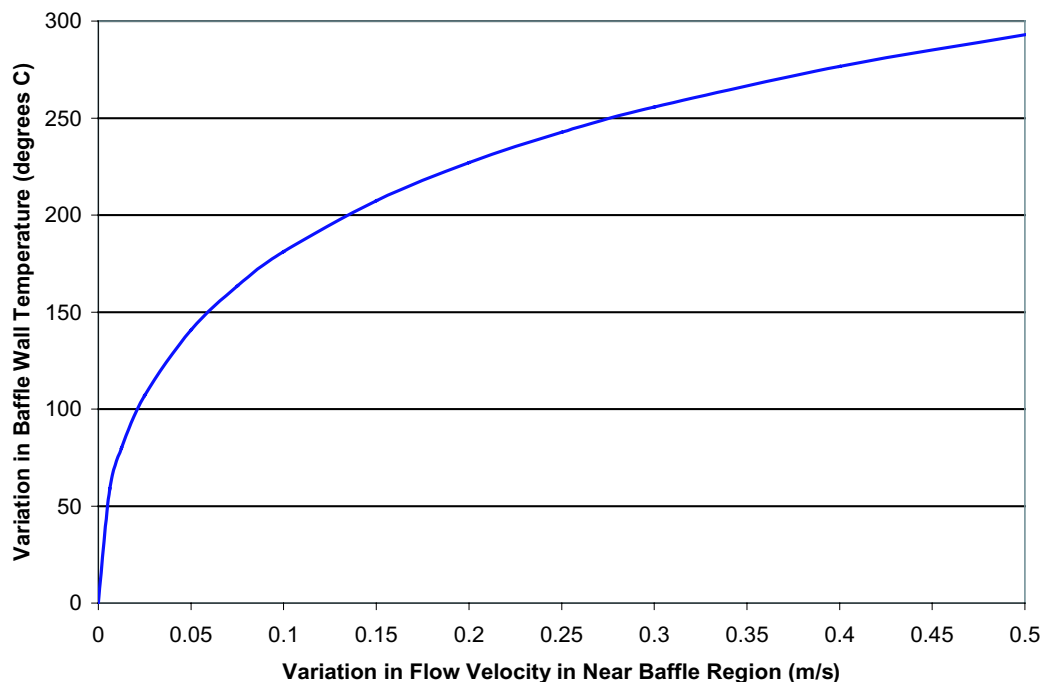


Figure 11.5 – Variation in the surface temperature of the flow baffle as a result of variation in the flow velocity.

surface temperature resulting from fluctuations in the flow velocity is approximately 290 °C.

Overall Variation in the Surface Temperature of the Flow Baffle

Clearly Equation 11.3 shows that the total variation in the flow baffle surface temperature is the sum of the fluctuations resulting from the variation in the flow velocity and the variation in the bulk fluid temperature. The total variation in the flow baffle surface temperature is shown in Figure 11.6. Since the maximum possible fluctuation in the bulk fluid temperature is about 110 °C and the maximum fluctuation in the flow velocity is about 0.5 m/s, the maximum possible fluctuation in the surface temperature is roughly 390 °C. It is important to note that this is only an estimate of the magnitude of the maximum possible instantaneous fluctuation in the surface temperature at a single point on the surface of the flow baffle resulting from the turbulent nature of the flow field. The fluctuation could be positive or negative and is not indicative of any change in the time averaged temperature distributions expected in the SNS mercury target.

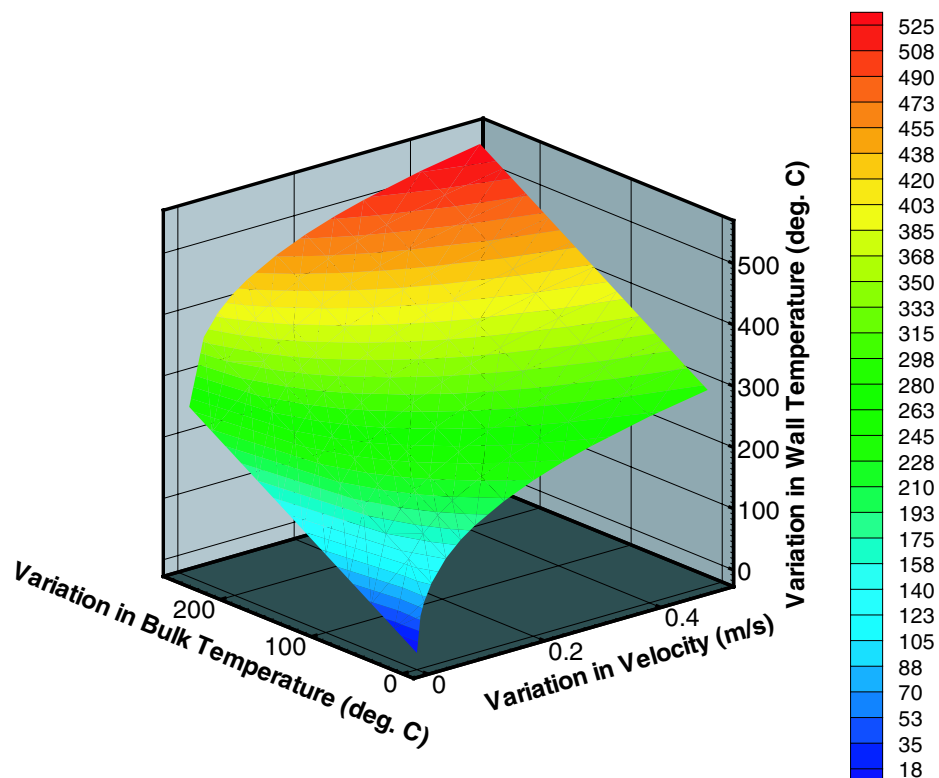


Figure 11.6 – Variation in the surface temperature of the flow baffle as a result of variations in the flow velocity and the bulk fluid temperature.

Chapter 12 Qualification of the Ultrasonic Doppler Profilometer

While Laser Doppler Velocimetry (LDV) is a relatively new technique for the measurement of flow velocity, its use is well documented and systems are commonly purchased as an off-the-shelf item. The Ultrasonic Velocity Profilometer (UVP) is not as well established as a reliable technique for time-dependent or even time-averaged velocity measurements. The use of the UVP in this study provides an opportunity to evaluate its abilities before it is used as the primary instrument in future prototypic mercury flow experiments. The time-averaged velocity data collected in the WTHL using the UVP instrument is compared to the data collected using the LDV instrument in Figures 12.1 and 12.2. The data shown in these figures is representative of the time-averaged velocity distribution in the WTHL test section at half of the nominal flow rate. The half-nominal flow rate was used so that

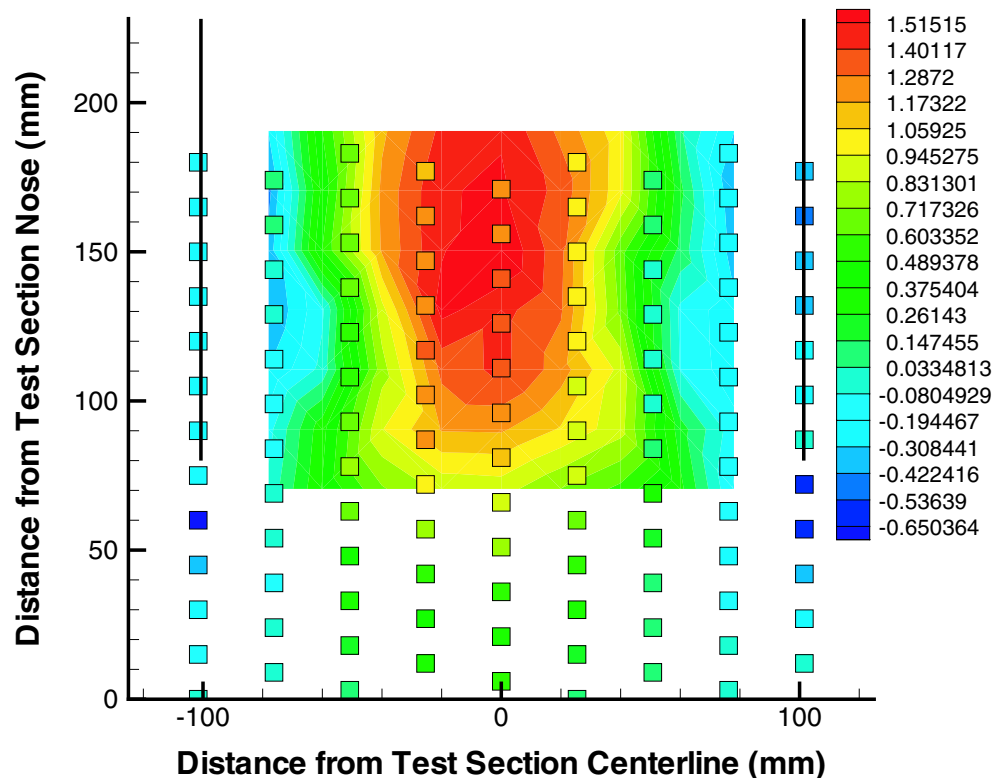


Figure 12.1 Comparison of LDV measurements and selected UVP measurements in the nominal WTHL test section. LDV values are shown in the contour plot, and UVP values are shown as point measurements.

the maximum and minimum velocity magnitudes would be within the measurable range for the UVP. The UVP measurements are typically within 15 percent of the nearest LDV measurements, which is comparable to the typical agreement between individual LDV measurements. Thus the UVP instrument provides an acceptable method for evaluation of velocity distributions in future systems.

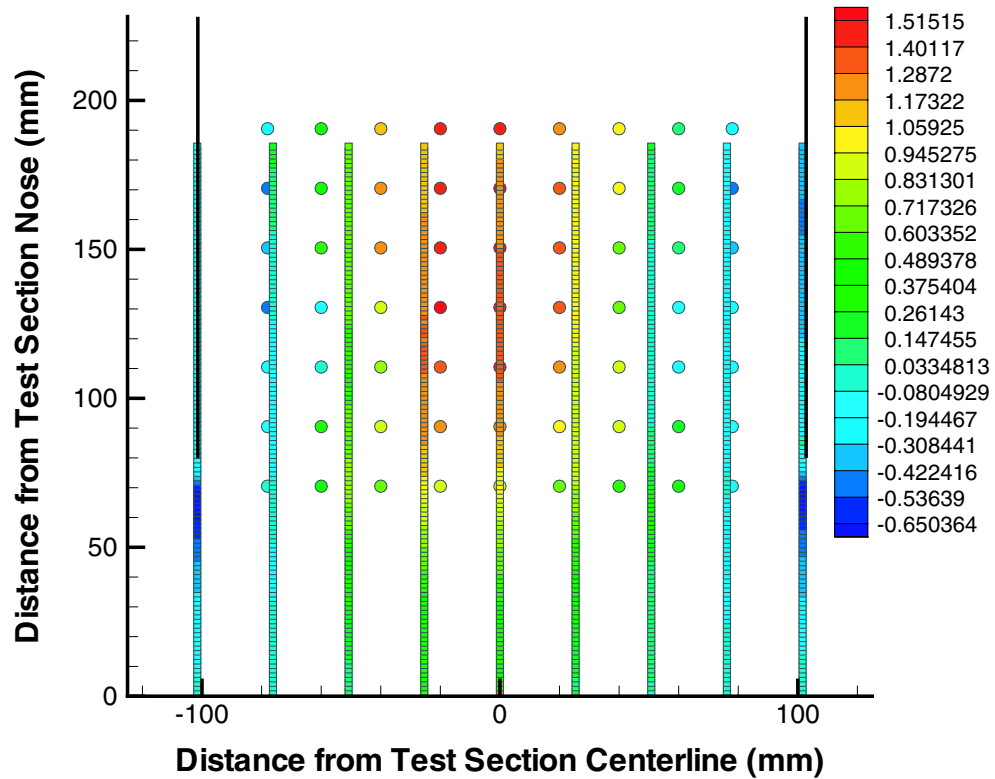


Figure 12.2 Comparison of LDV measurements and all UVP measurements in the nominal WTHL test section. LDV values are shown as circular data points, and UVP values are shown as square data points.

Chapter 13 Validation of the CFD Simulation of the SNS Mercury Target

The primary purpose of the SNS thermal hydraulic experiment program is to provide data for the validation of the Computational Fluid Dynamics (CFD) model. Validation of a model by qualitative and quantitative experiments identifies the shortcomings of the model and provides a basis for the determination of whether the errors in the model are conservative. A conservative error leads to the calculation of a value of some parameter that indicates that the parameter is closer to the engineering constraints than it would be in the actual system. A non-conservative error has the opposite effect on the parameters of interest, indicating that they are farther from the engineering constraints than they really are.

13.1 Comparison of CFD Simulation to Flow Visualization Studies in the SNS Thermal Hydraulic Experiments

Flow visualization studies in the University of Tennessee Air Test Facility (UTATF) and the Water Thermal Hydraulic Loop (WTHL) provide a qualitative evaluation of the general flow behavior in the SNS mercury target. These experiments indicate that two primary recirculation zones develop in the exit channel near the leading edge of each flow baffle as predicted by the CFD simulation. The third recirculation zone predicted to develop in the nose region of the target by the simulation does not appear to develop as a stable flow structure in the UTATF and WTHL experiments. Both experiments indicate that the flow tends to alternate between symmetric and asymmetric modes in a pseudo-periodic fashion. While the simulation does not predict this behavior directly, it does indicate that multiple steady state solutions may exist. Further flow visualization studies in the WTHL indicate that a vortex shedding phenomenon occurs in the downstream half of the exit channel. Neither steady state nor transient simulation using the CFD model predict such behavior. The flow behavior predicted by the CFD model and the behavior observed in the WTHL experiments are shown in Figures 13.1 and 13.2 for comparison.

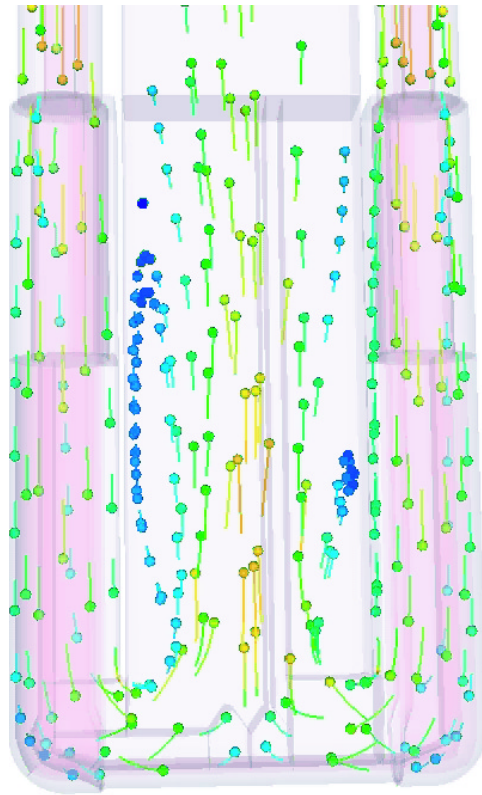


Figure 13.1 – Streamlines predicted by the CFD simulation of the nominal WTHL test section at half nominal flow (Wendel, 2001 - Used with Permission). Click on the image to activate the animation.



Figure 13.2 – Flow visualization experiment using bubble injection in the nominal WTHL test section at half nominal flow. To see video footage of the experiment, click on the image to activate animation.

13.2 Comparison of Velocity Fields Predicted by the CFD Simulation and Measured in the WTHL

Velocity field maps developed from LDV measurements in the WTHL provide more qualitative data for the evaluation of the predictive capabilities of the CFD simulation. In general, the velocity measurements in the WTHL indicate that the time-averaged flow behavior in the test section is very similar to the simulation. The boundaries of the two stationary recirculation zones are easily identified by the reduction in flow velocity and increase in turbulence intensity in those regions. The flow between the two recirculation zones develops a high velocity jet that is not as turbulent as the flow in the recirculation zones. The time averaging washes out the time-dependent effects, such as the vortex shedding phenomena and the pseudo-periodic oscillation between modes. Thus, the time-averaged behavior appears relatively symmetric, as predicted by the preliminary CFD simulations.

For direct comparison, the measured and predicted velocities are shown plotted together in Figure 13.3. The velocity magnitudes shown are indicative of the velocity profiles at the horizontal midplane in the nominal WTHL test section at the nominal flow velocity. Both the magnitude and direction of the predicted velocities show good agreement with the measured values, typically within 10 percent in the primary flow regions. The predicted velocities in the recirculation zone regions are slower than the measured values, and the predicted direction do not agree as well as in the jet region of the exit channel.

Figures 13.4 and 13.5 show the measured and predicted values for the axial and cross-stream components of the turbulence intensity. The predicted values are the same in both directions because of the inherent isotropy in the turbulence model used in the simulation. The cross-stream components of the turbulence intensity show reasonable agreement, although the code may have a tendency to under predict this component in the region upstream of the flow baffles and over predict it in the region downstream of the baffles. The axial components of the turbulence intensity show good agreement in the region upstream of the flow baffles and in the recirculation zones, but good agreement is not achieved in the central jet region of the exit channel flow.

As seen in previous chapters, statistical fluctuations in the flow velocity in the WTHL test section resulting from inherent turbulence in the flow field lead to variations in the measured velocity between LDV data sets. Individual velocity data sets at selected distances from the test section nose are plotted along

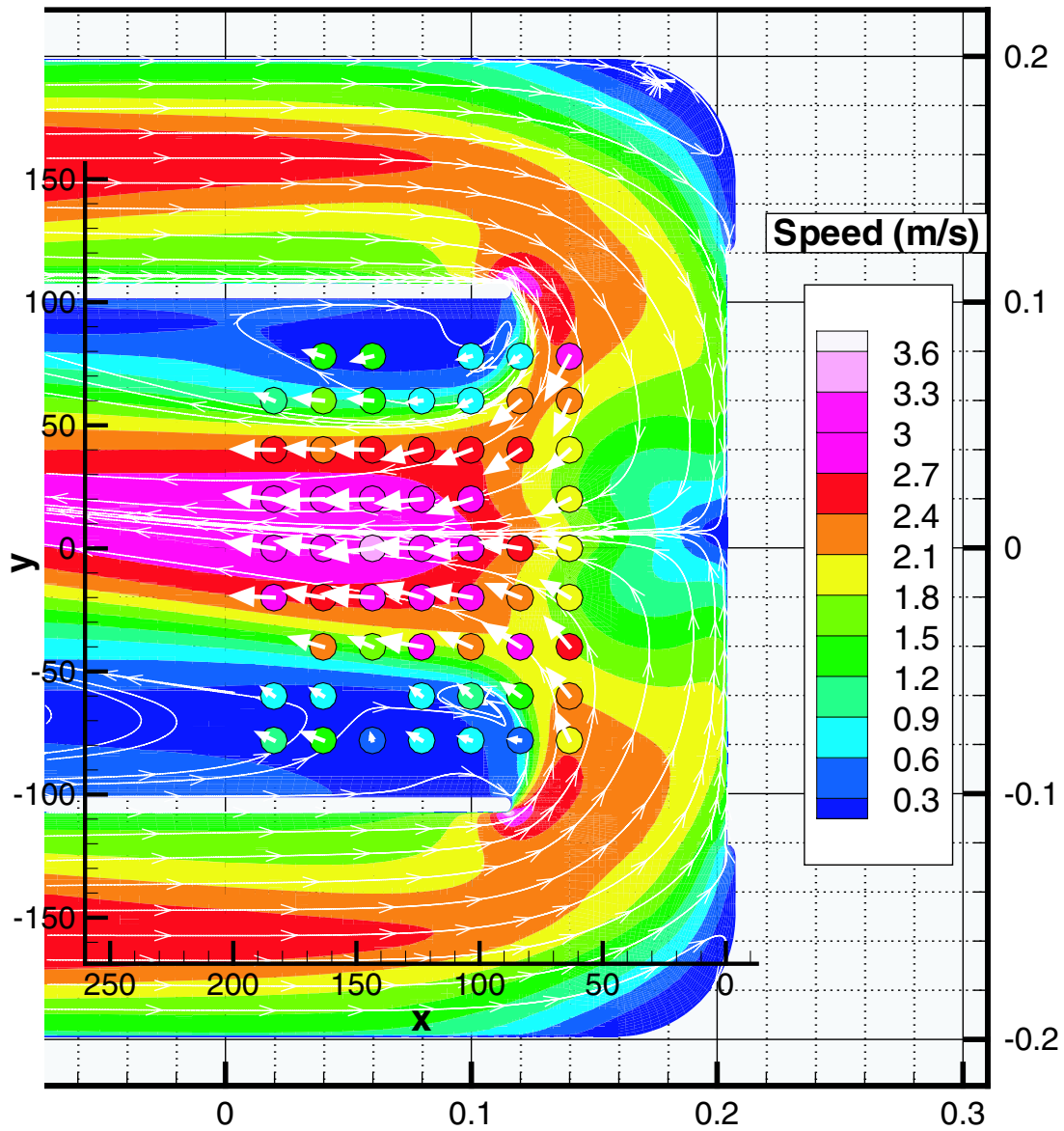


Figure 13.3 – Comparison of LDV velocity measurements in the WTHL test section at nominal flow with predicted values from the CFD model (Wendel, 2001 - Used with Permission). Predicted values are shown as a contour plot. Measured values are shown as individual data points.

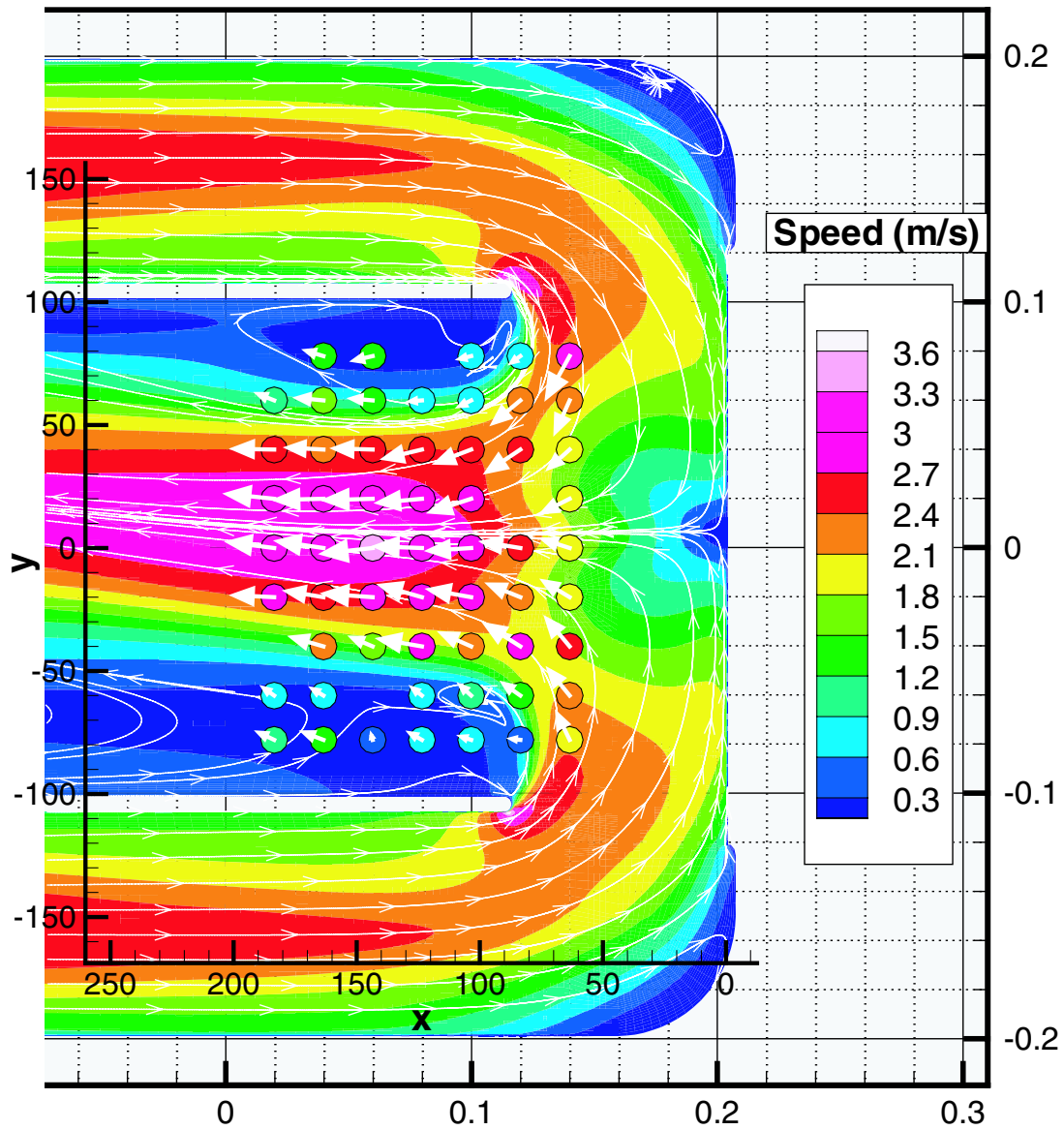


Figure 13.4 – Comparison of the axial component of the turbulence intensity from LDV measurements in the nominal WTHL test section at the nominal flow velocity and CFD simulation (Wendel, 2001 - Used with permission). The predicted values are shown as a contour plot. Measured values are shown as individual data points.

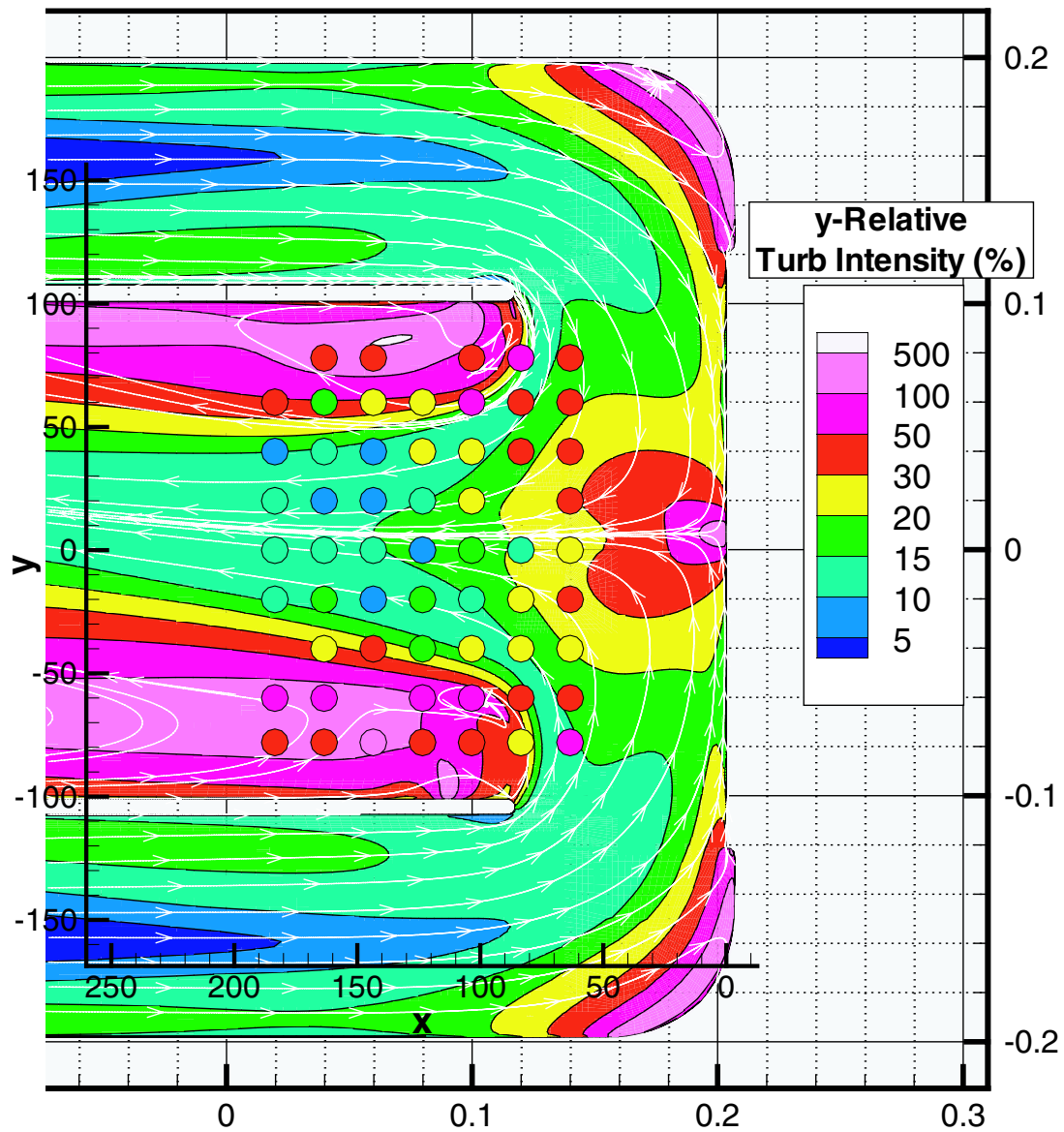


Figure 13.5 – Comparison of the cross-stream component of the turbulence intensity from LDV measurements in the nominal WTHL test section at the nominal flow velocity and CFD simulation (Wendel, 2001 - Used with Permission). The predicted values are shown as a contour plot. Measured values are shown as individual data points.

with predicted values on separate linear axes in Figure 13.6. Individual measurements of the cross-stream component of the turbulence intensity are plotted in a similar manner in Figure 13.7. These figures indicate that the variation between data sets is typically less than the variation between the measured and predicted values. Thus, the discrepancies between the measured and predicted time-averaged values can be considered real differences rather than an effect of the inherent stochastic nature of the flow field.

Closer examination of the predicted and measured velocities in the recirculation zone region near the leading edge of the flow baffle offers a possible explanation for the difference in the measured and predicted values of the velocity in those regions. As seen in Figure 13.8, the measured data appears to indicate that the simulation dramatically over predicts the size of the recirculation zone. The over prediction of the size of the recirculation zone could be related to the under prediction of the axial component of the turbulence intensity in the central jet region of the exit channel. The inability of the code to predict large-scale turbulent structures, such as the vortex shedding structures seen in the flow visualization experiments, could also lead to the over prediction of the size of the recirculation zones. Both of these characteristics of the flow field would impact the momentum or vorticity equations used by the code to develop the streamline predictions.

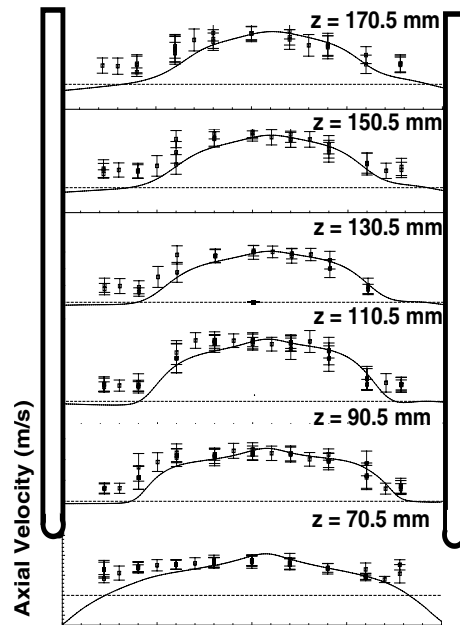


Figure 13.6 – Comparison of multiple LDV velocity data sets and predicted velocity values (Wendel, 2001 - Used with Permission). Solid lines indicate predicted values. Individual data points indicate measured values.

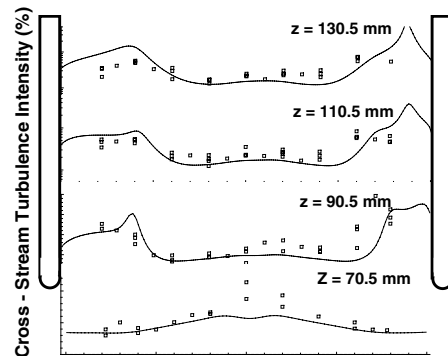


Figure 13.7 – Comparison of multiple LDV cross-stream turbulence intensity data sets and predicted values (Wendel, 2001 - Used with Permission). Solid lines indicate predicted values. Individual data points indicate measured values.

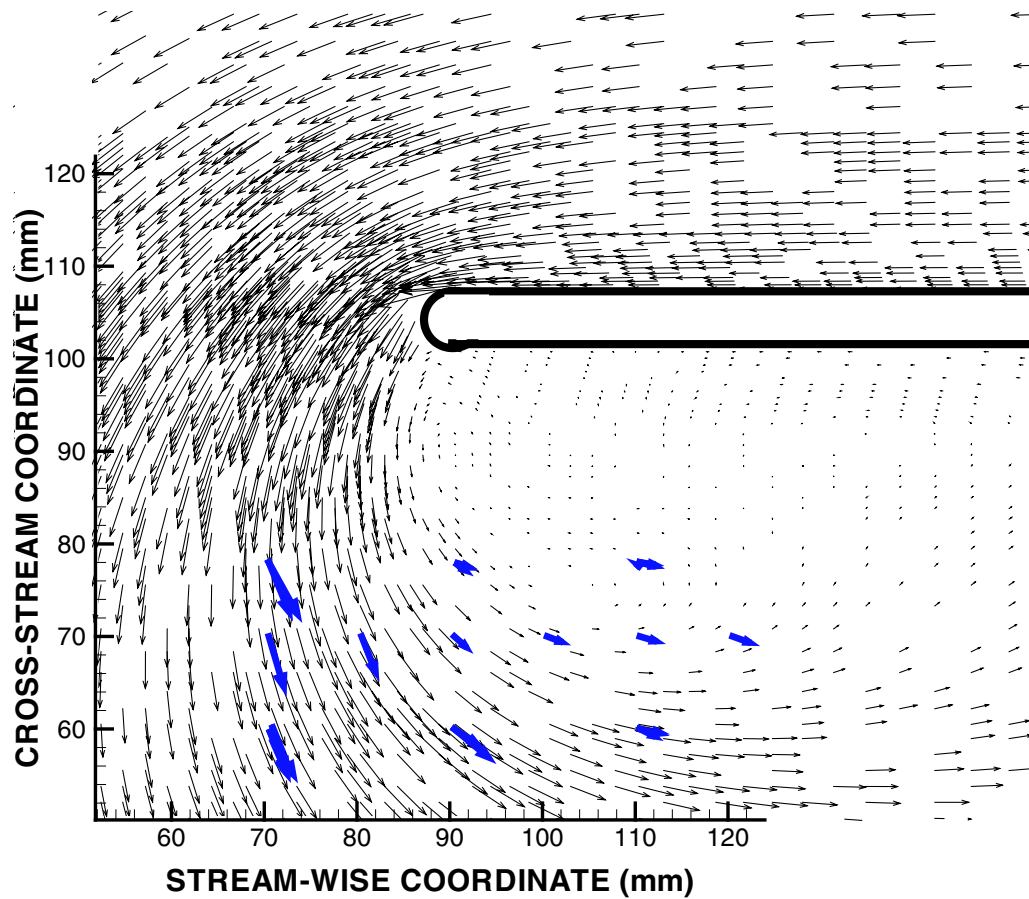


Figure 13.8 – Close-up view of predicted and measured velocity vectors in the recirculation zone region near the leading edge of the flow baffle (Wendel, 2001 - Used with Permission). Small black arrows indicate predicted vectors. Bold blue arrows indicate measured vectors.

13.3 Conservative Versus Non-conservative Errors

Based on the correlation between the observed behavior in the SNS thermal hydraulic experiments and the CFD predictions, the CFD simulation provides a conservative approximation to the time-averaged behavior of the SNS mercury target. The simulation generally captures the typical behavior of the flow field. The simulation tends to over predict the size of the primary recirculation zones and under predict the velocity in those regions. The code also tends to under predict the axial component of the turbulence intensity in much of the exit channel. The errors in the prediction of the recirculation zone size and speed tend to artificially increase the residence time of a fluid in the recirculation, encouraging the over prediction of the maximum temperature in those regions. The error in the prediction of turbulence artificially reduces turbulent mixing and dissipation in the exit channel

flow. The under prediction of turbulence intensity leads to the over prediction of temperature gradients in the flow field. One would expect the combination of these three errors to lead to the over prediction of the critical fluid and structural temperatures, indicating that the CFD model is conservative for use in the design of the SNS mercury target.

Chapter 14 Conclusion

The primary mission of the Spallation Neutron Source (SNS) thermal hydraulic experiments is the development of a reliable data set for validation of the computational model used in the design of the SNS mercury target. The Computational Fluid Dynamics (CFD) simulation will provide the primary thermal hydraulic evaluations for the design of the initial mercury target as well as the development of future modifications. Therefore, it is critical that the model accurately, or at least conservatively, predicts the velocity and temperature distributions in the mercury target.

Two scaled experiments are used in this study to characterize the behavior of the flow field in the SNS target. The first facility, the University of Tennessee Air Test Facility (UTATF), is a scaled representation of the SNS target that uses air as a surrogate fluid for liquid mercury. The facility is not scaled to provide an exact representation of the SNS target flow. Thus, it is used primarily to identify phenomena of interest for further study in the other experiments. The second facility, the Water Thermal Hydraulic Loop (WTHL), is a scaled experiment that uses water as a surrogate fluid for the liquid mercury. The WTHL test section is an exact replica of the SNS target design, and the flow loop is scaled to match the flow rate and turbulence intensity in the mercury target. This facility provides high fidelity data for direct validation of the predictions of the computational simulation.

Flow visualization experiments provide qualitative data about the flow field through direct observation of flow structures and patterns. Flow visualization studies in both facilities reveal that the flow is highly diffusive and quite turbulent in nature. Two primary recirculation zones develop in the exit channel near the leading edge of each flow baffle. The remainder of the fluid forms a central jet between these two zones. In both facilities, a pseudo-periodic oscillation between symmetric and asymmetric modes is observed. In the WTHL experiments, large-scale turbulent structures similar to the vortex shedding phenomena in the wake of an object in cross-flow develop in the exit channel downstream of the recirculation zones. The development of these structures could be linked to the observed pseudo-periodic oscillations.

More quantitative studies are performed in the nominal WTHL test section using Laser Doppler Velocimetry (LDV) to make local velocity measurements at various

positions in the flow field. From these measurements, a three-dimensional map of both the velocity and turbulence intensity can be developed. These studies confirm the general time-averaged flow behavior seen in the flow visualization experiments. Repeated LDV measurements in the WTHL test section indicate that time-averaged velocity measurements are repeatable within 10 percent regardless of the position in the flow field.

Additional velocity measurements are made in the WTHL test section using an Ultrasonic Velocity Profiliometer (UVP). The UVP can easily be used to measure velocities in regions of the flow field that cannot be reached by LDV because of restrictive optical properties in the test section. However, the UVP technology is not as well proven as the LDV instrument. Therefore, the primary purpose of this component of the study is to establish the credibility of the UVP instrument for future prototypic mercury experiments in which other methods are not available. The UVP data collected in the nominal WTHL test section shows excellent agreement with the LDV data collected under the same conditions. Indeed, the difference between the two data sets is similar to the difference between individual LDV data sets in the repeatability studies.

All of the data collected in the two experimental facilities ultimately contributes to the validation of the computational model. The general flow behavior predicted by the model is generally replicated by the experiments. On a time-averaged basis the flow field is relatively symmetric with two large recirculation zones in the exit channel near the leading edge of each flow baffle. Flow visualization studies indicate pseudo-periodic oscillation between flow modes, possibly triggered by a vortex shedding phenomenon. While transient versions of the model do not predict this behavior, steady state calculations appear to indicate multiple symmetric and asymmetric solutions. Comparison with quantitative data indicates that the computational model provides a conservative prediction of the velocity and temperature profiles within the SNS mercury target.

Chapter 15 Recommendations for Future Work

The third and final Spallation Neutron Source (SNS) thermal hydraulic experiment will be completed in the summer of 2001. The Target Test Facility (TTF) is a prototypic study of the mercury target and the associated mercury systems. In addition to thermal hydraulic testing the facility will be used to test control systems and remote handling equipment for maintenance and target vessel replacement.

The thermal hydraulic components of the study should focus on confirmation of observed behavior and identification of phenomena that can not be observed in other facilities rather than comprehensive evaluation of the flow field. While many heat transfer and fluid property experiments are planned in the facility, the UVP instrument used in this study will be available for examination of the flow patterns in the prototypic mercury experiment. Key issues that should be examined include:

- confirmation of the scaling of the scaled experiments by measurement of turbulence intensity
- evaluation of the size of the recirculation zone in the mercury test section
- confirmation of oscillation between symmetric and asymmetric modes
- evaluation of the stability of the inlet flow rates versus the inlet pressure drop

The completion of these four studies would provide sufficient confirmation of the results presented in this study to conclude that the simulation provides a conservative approximation to the actual velocity field. Additional data regarding the thermodynamic properties of the liquid mercury and the likelihood of non-wetting between the liquid mercury and surface of the target structure is needed before the temperature profiles predicted by the simulation can be thoroughly confirmed. Heated experiments in the TTF and a separate experiment to examine heat transfer coefficients and fluid properties in a narrow heated channel will provide the data needed to make this confirmation.

References

- Crane, R.I., and J. Sabzvari, "Heat Transfer Visualization and Measurement in Unstable Concave-Wall Laminar Boundary Layers," *Journal of Turbomachinery*, **111**, 1989.
- Duncan, W.J., A.S. Thom, and A.D. Young, *Mechanics of Fluids*, American Elsevier Publishing Company, Inc., 1970
- Floryan, J.M., "On the Coertler Instability of boundary Layers," *Progress in Aerospace Science*, **28**, 1991.
- Han, J. C., P. R. Chandra, S. C. Lau, "Local Heat/Mass Transfer Distributions Around Sharp 180 Degree Turns in Two-Pass Smooth and Rib Roughened Channels," *Journal of Heat Transfer*, **110**, February 1988.
- Heyreichs, K., and A. Pollard, "Heat Transfer in Separated and Impinging Turbulent Flows," *International Journal of Heat and Mass Transfer*, **39**, no. 12, 1996.
- Hirota, M., et al., "Heat/Mass Transfer Characteristics in Two-Pass Smooth Channels with a Sharp 180-deg Turn," *International Journal of Heat and Mass Transfer*, **42**, 1999.
- Incropera, F. P., and D.P. DeWitt, *Fundamentals of Heat and Mass Transfer*, John Wiley and Sons: New York, 1990.
- Kinsler, J.E., and A.R. Frey, *Fundamentals of Acoustics*, John Wiley and Sons: New York, 1962.
- "Laser Doppler Velocimetry Catalog: Overview," TSI Home Page, www.tsi.com/fluid/ldv_cat/overview.htm, May 2000.
- Liou, T. M., Y.H. Hwang, L. Chen, "Prediction of Confined Three-Dimensional Impinging Flows with Various Turbulence Models," *Journal of Fluids Engineering*, **114**, June 1992.
- Marusik, I., A. Uddin, and A. Perry, "Similarity Law for the Streamwise Turbulence Intensity in Zero-Pressure-Gradient Turbulent Boundary Layers," *Physics of Fluids*, v. 9, no. 12, Dec 1997.
- Metzger, D. E., and M. K. Sahm, "Heat Transfer Around Sharp 180-deg Turn in Smooth Rectangular Channels," *Journal of Heat Transfer*, **108**, August 1986.
- Pedlosky, J., *Geophysical Fluid Dynamics*, 1985.
- Perry, A., and J. Li, "Experimental Support for the Attached-Eddy Hypothesis in Zero-Pressure-Gradient Turbulent Boundary Layers," *Journal of Fluid Mechanics*, v. 218, 1990.
- Perry, A., S. Henbest, and M. Chong, "A Theoretical and Experimental Study of Wall Turbulence," *Journal of Fluid Mechanics*, v. 165, 1986.
- Pointer, W. David, A. E. Ruggles, S. M. Chae, D. F. Cox, J. M. Crye, D. K. Felde, P. A. Jallouk, "Characterization of Flow Patterns in the Nominal SNS Mercury

- Target through Flow Experiments in the Water Thermal Hydraulic Loop (WTHL),” SNS/TSR-194, May 2000.
- Pointer, W. David, A. E. Ruggles, S. M. Chae, D. F. Cox, J. M. Crye, D. K. Felde, P. A. Jallouk, “Flow Characterization in the Water Thermal Hydraulic Loop (WTHL) for a Modified Test Section with Extended Flow Baffles and Additional Flow Controls,” SNS/TSR-196, August 2000.
- Pointer, W. David, A. E. Ruggles, S. M. Chae, D. F. Cox, J. M. Crye, D. K. Felde, P. A. Jallouk, “Summary of Water Thermal Hydraulic Loop (WTHL) Experiments for Characterization of Flow Behavior in the SNS Mercury Target,” SNS/TSR-205, September 2000.
- Reynolds, A. J., *Turbulent Flows in Engineering*, John Wiley & Sons: London, 1974.
- Roberts, P. H., and A. M. Soward, *Rotating Fluids in Geophysics*, Academic Press, 1978.
- Schetz, J. A., *Boundary Layer Analysis*, Prentice Hall: Englewood Cliffs, NJ, 1993.
- Spalart, P., “Direct Simulation of a Turbulent Boundary Layer Up to $Re_q = 1410$,” *Journal of Fluid Mechanics*, v. 187, 1988.
- “Spallation Neutron Source: Advancing the Frontiers of Science and Technology”, pamphlet, SNS Project Office: Oak Ridge, TN, 2000.
- Spallation Neutron Source Design Manual*, SNS Project Office: Oak Ridge, TN, September 1998.
- Townsend, A. A., *The Structure of Turbulent Shear Flow*, Second Edition, Cambridge University Press: Cambridge, 1976.
- Wendel, Mark, Personal Communication, February 2001.
- Wendel, Mark, “SNS Target Thermal-Hydraulic Design and Analysis,” International Workshop on Mercury Target Development for Pulsed Spallation Neutron Sources, Japan Atomic Energy Research Institute (JAERI), November, 2000.
- White, F. M., *Viscous Fluid Flow*, Second Edition, McGraw-Hill: Boston, 1991.
- Zhukauskas, A., and J. Zhiugzhda, *Heat Transfer of a Cylinder in Cross-Flow*, Hemisphere Publishing Corporation: Washington, 1985.

Appendix

Additional Velocity Maps from WTHL LDV Measurements

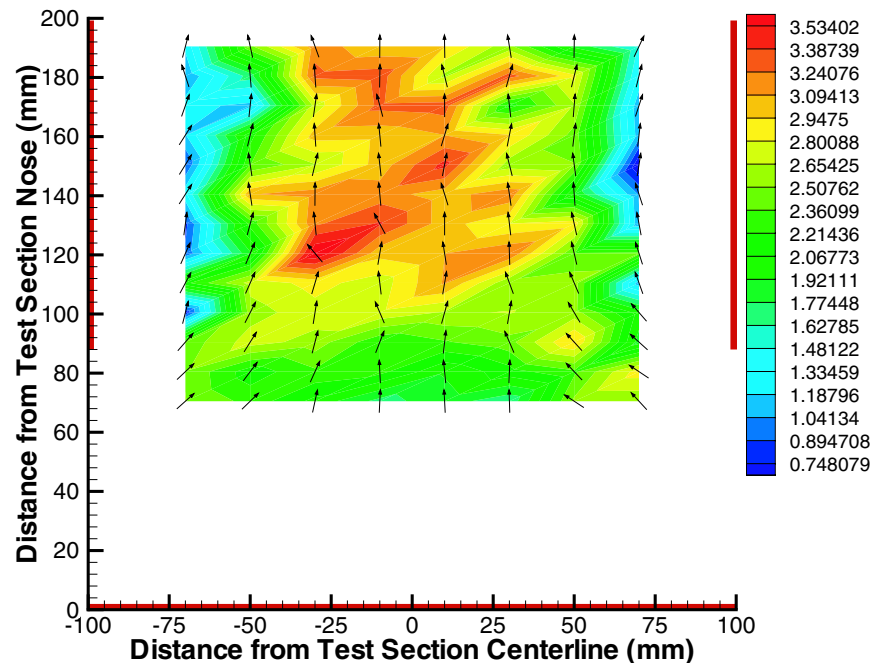


Figure A.1 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the upper inside surface of the test section.

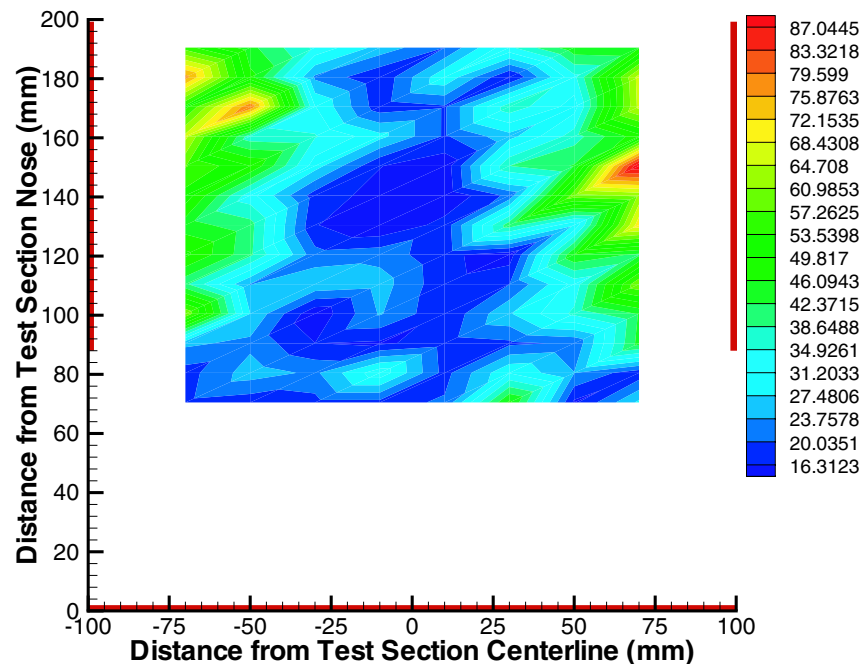


Figure A.2 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the upper inside surface of the test section.

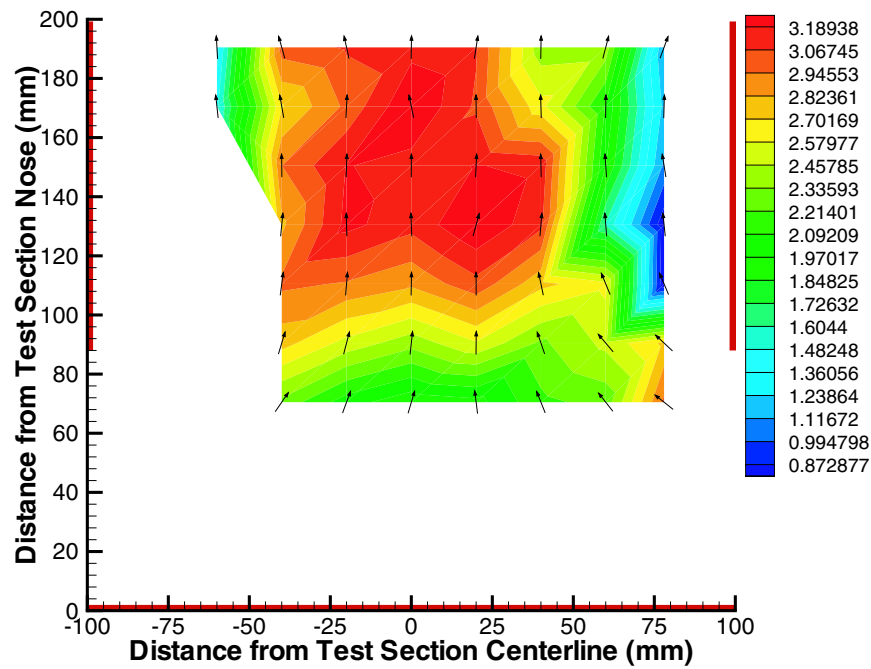


Figure A.3 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the upper inside surface of the test section.

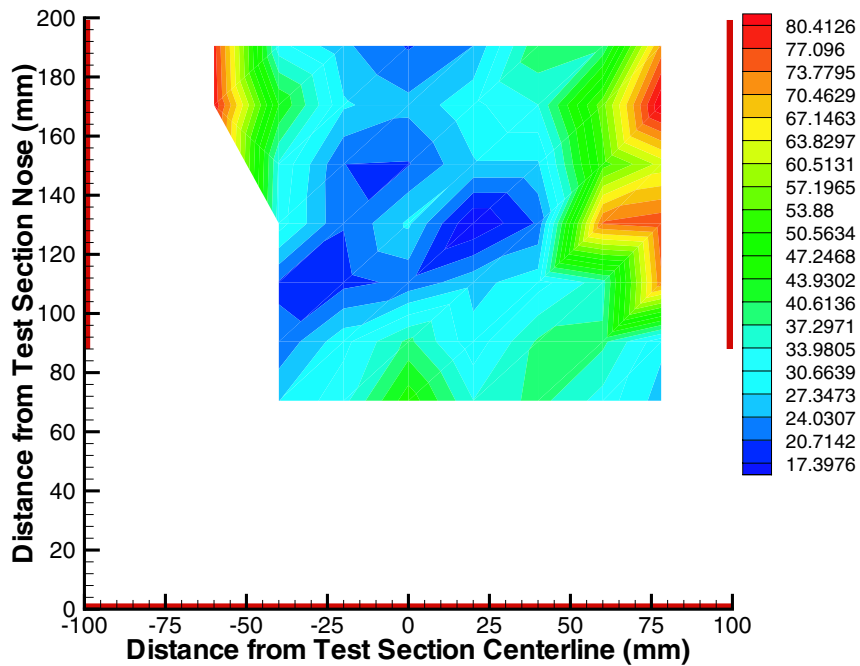


Figure A.4 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the upper inside surface of the test section.

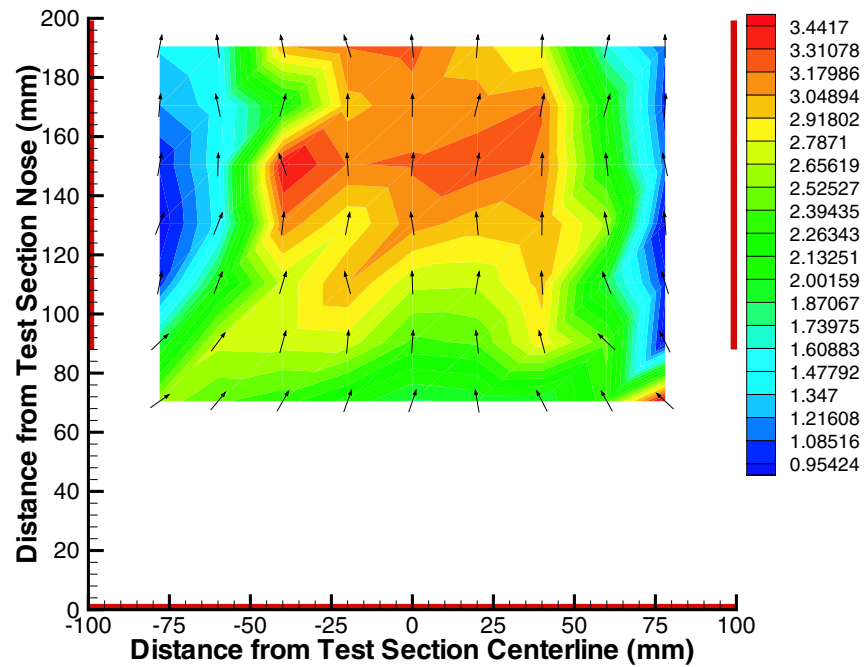


Figure A.5 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the upper inside surface of the test section.

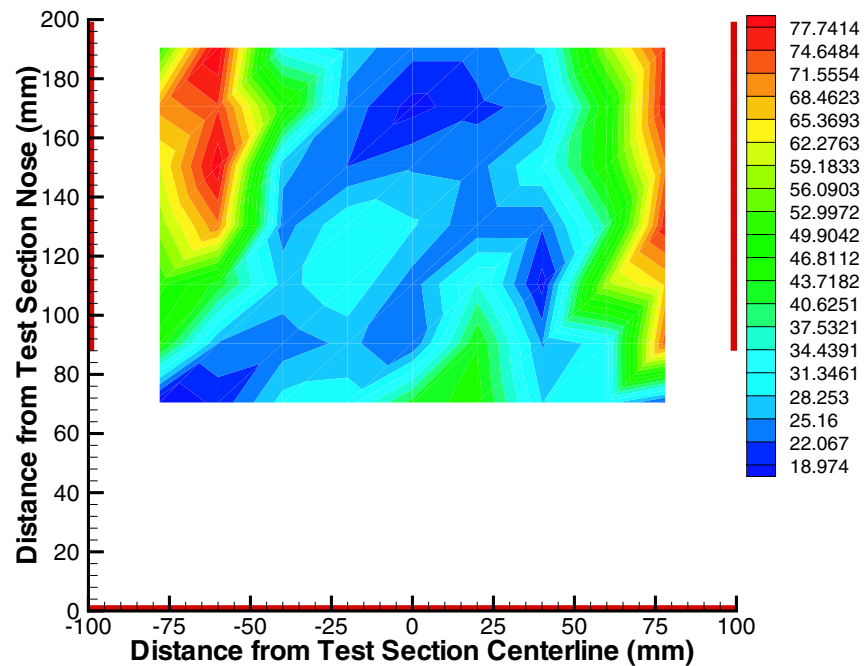


Figure A.6 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the upper inside surface of the test section.

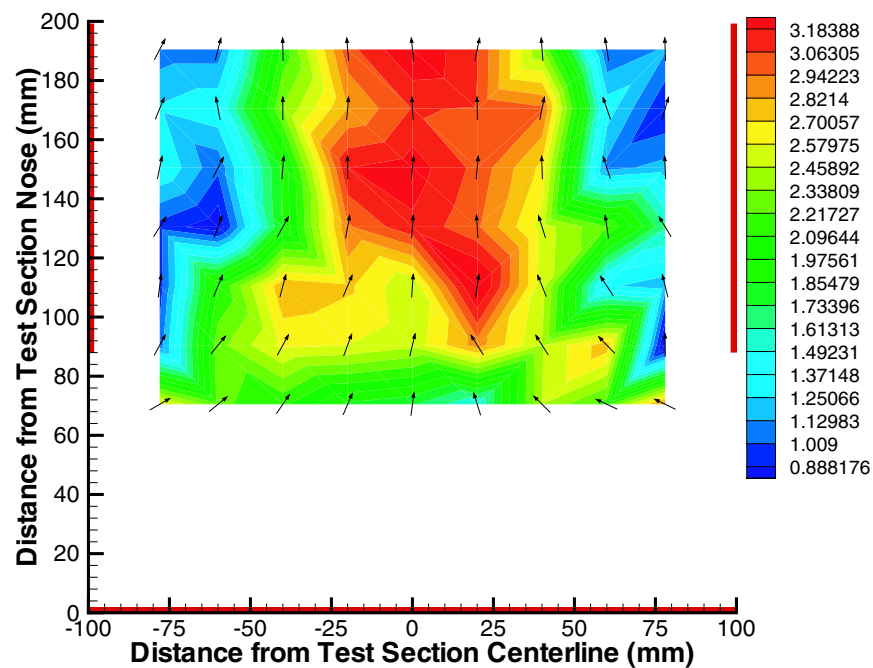


Figure A.7 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 20 mm from the upper inside surface of the test section.

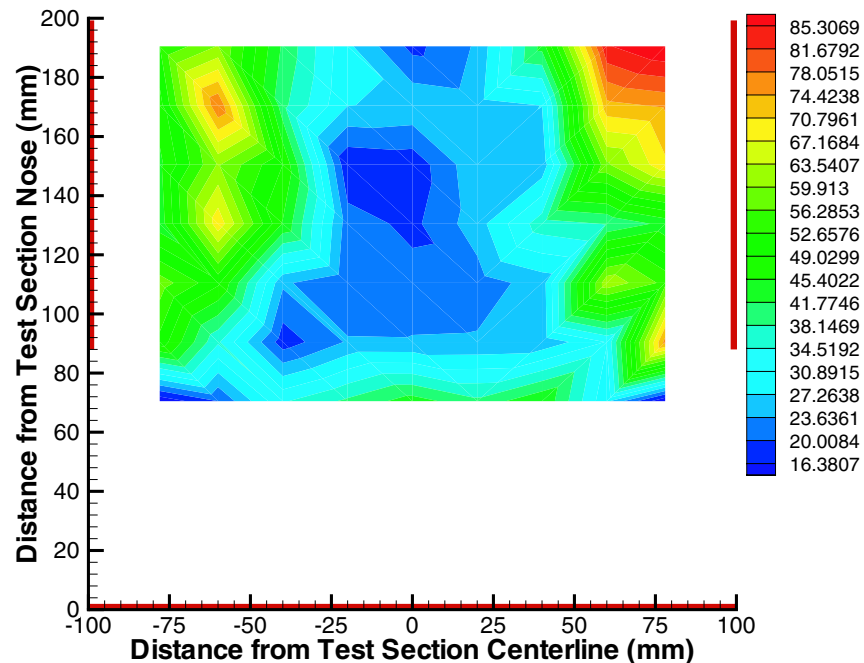


Figure A.8 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 20 mm from the upper inside surface of the test section.

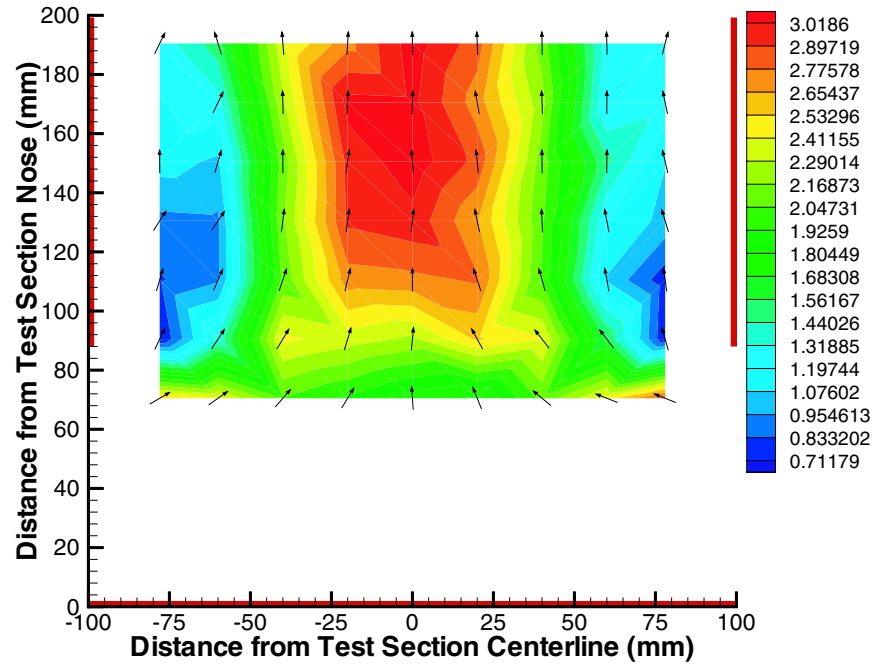


Figure A.9 Velocity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.

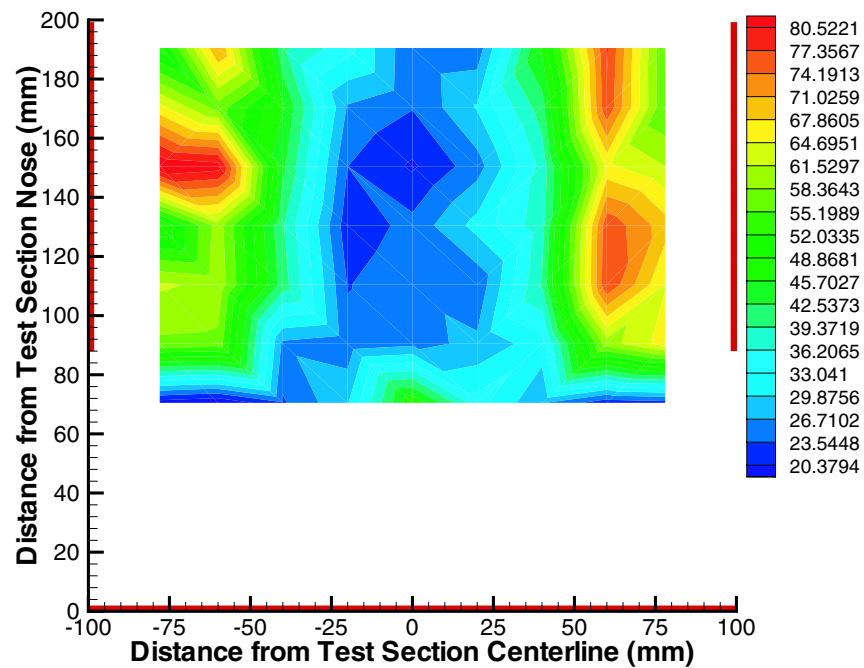


Figure A.10 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.

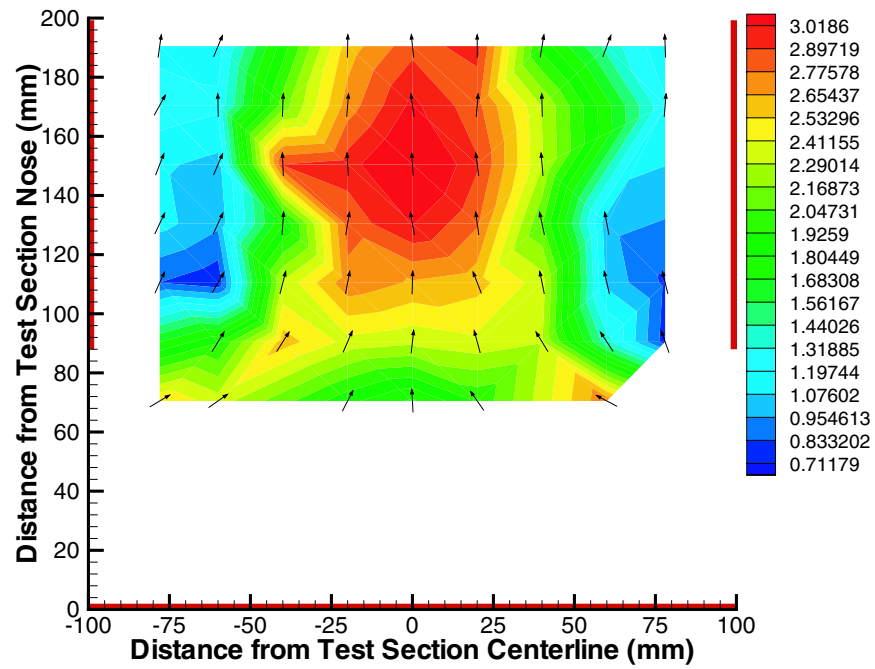


Figure A.11 Velocity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.

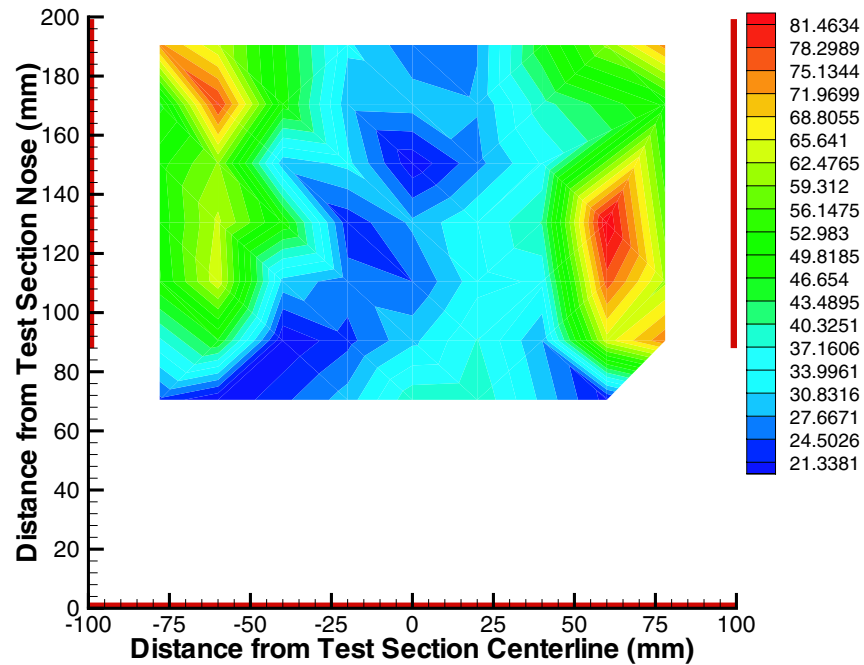


Figure A.12 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.

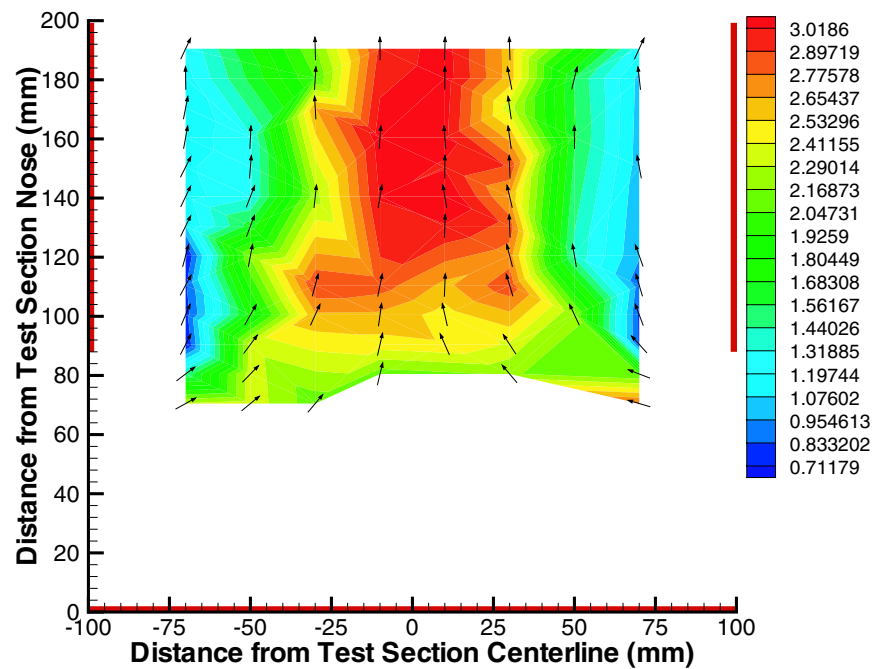


Figure A.13 Velocity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.

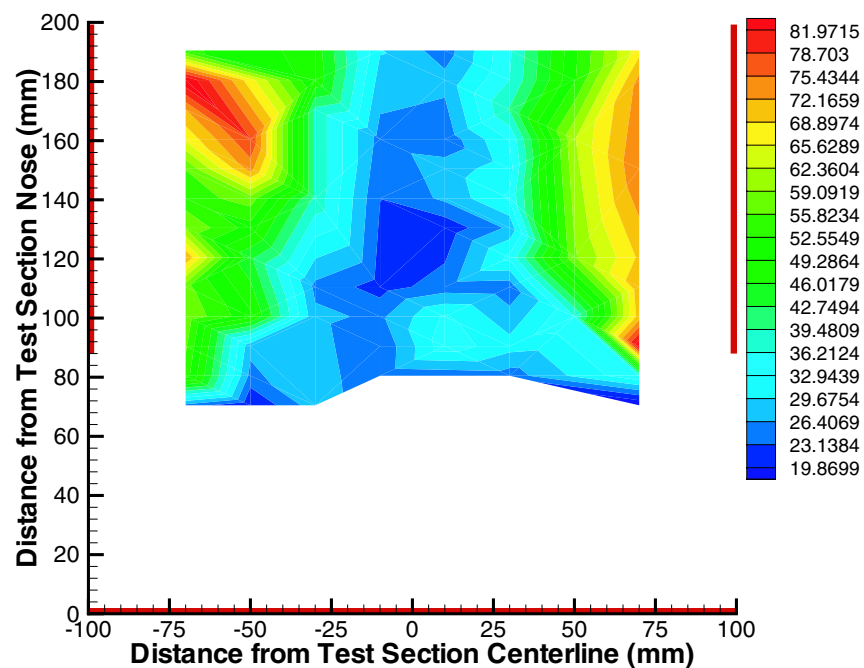


Figure A.14 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.

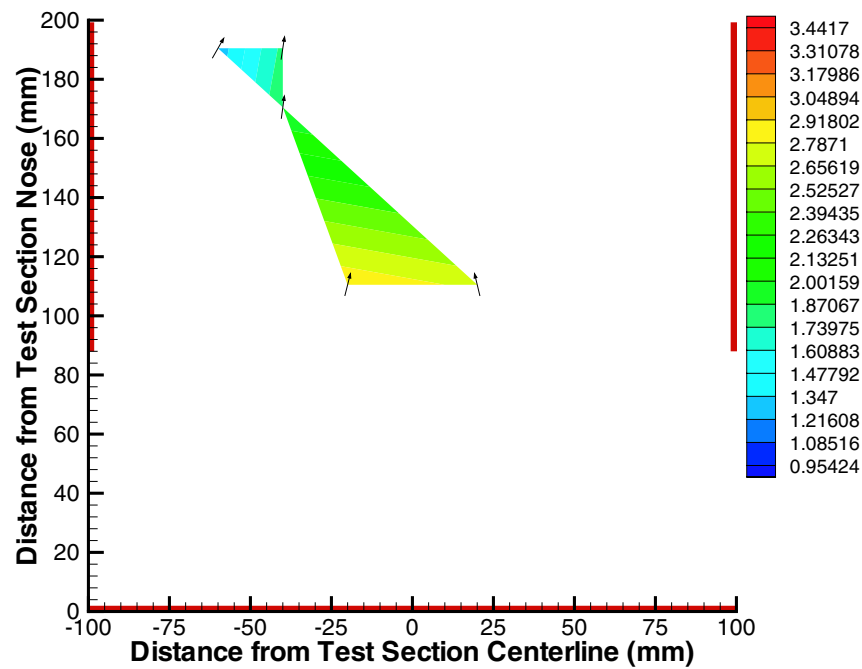


Figure A.15 Velocity measured in the nominal WTHL test section at nominal flow rate at the test section centerline. This data set is incomplete due to rust buildup in the loop process water.

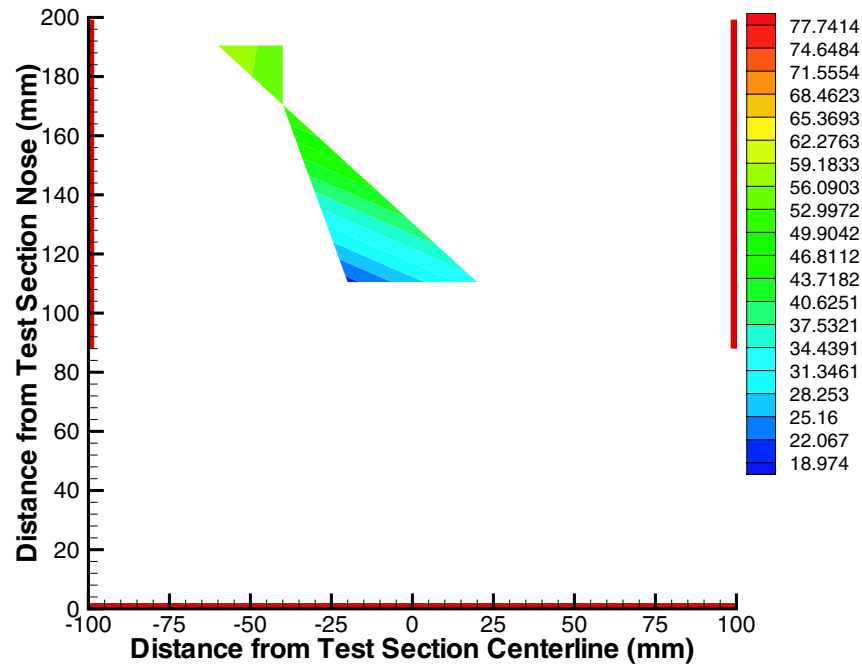


Figure A.16 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at the test section centerline. This data set is incomplete due to rust buildup in the loop process water.

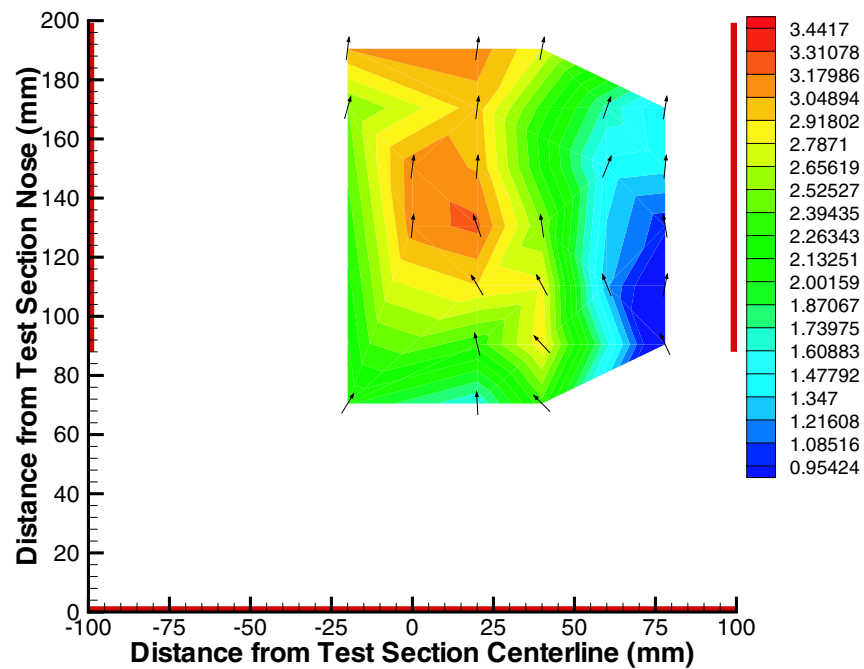


Figure A.17 Velocity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.

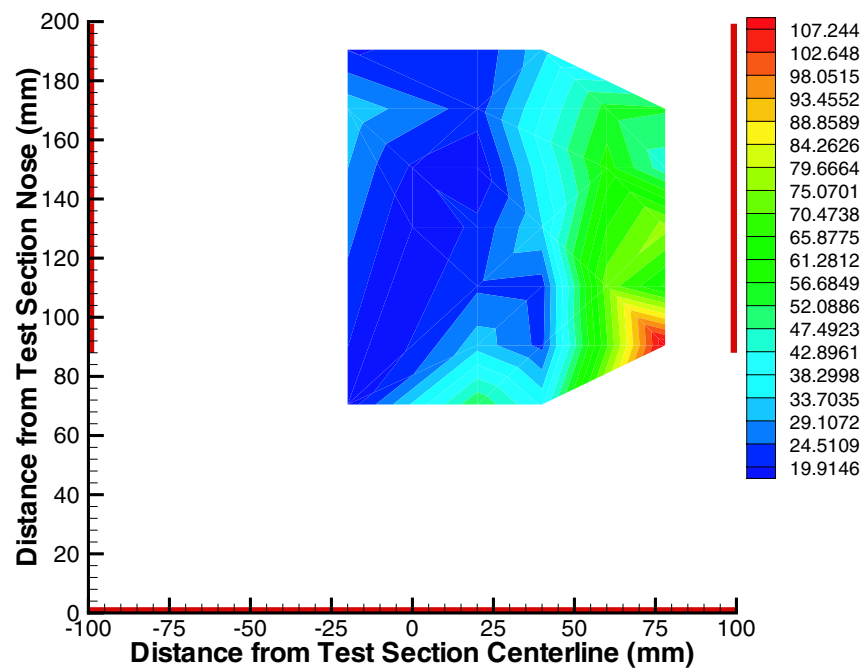


Figure A.18 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.

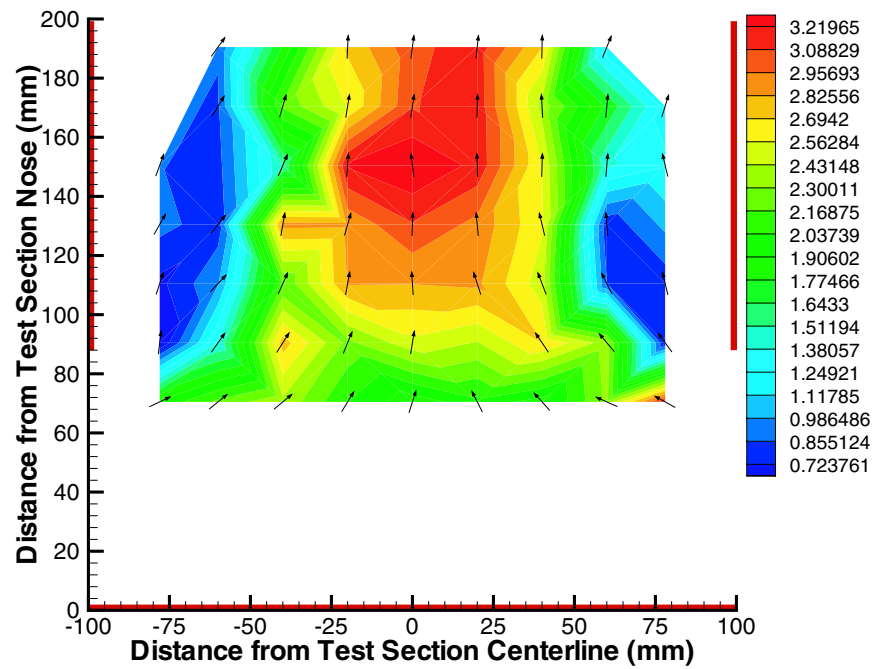


Figure A.19 Velocity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.

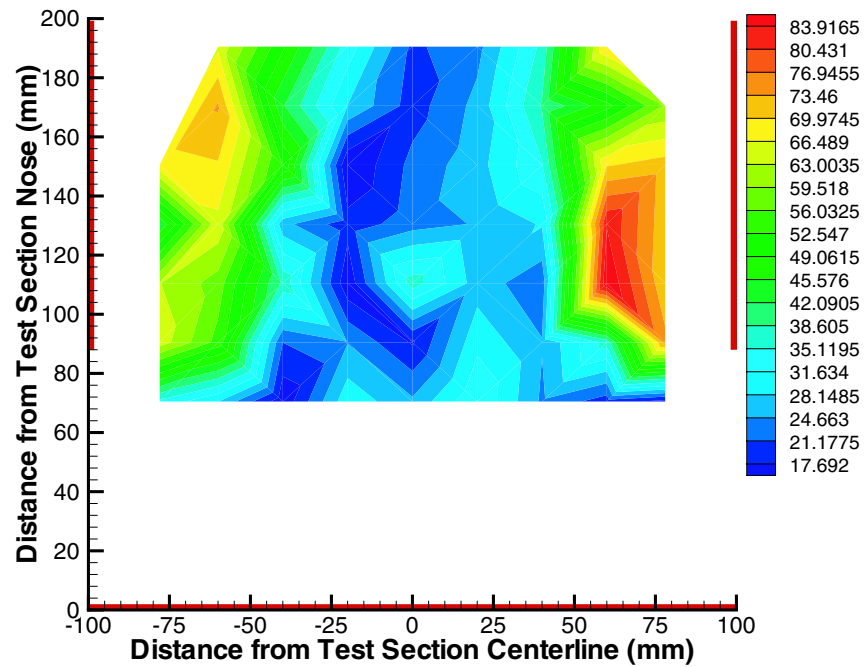


Figure A.20 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at the test section centerline.

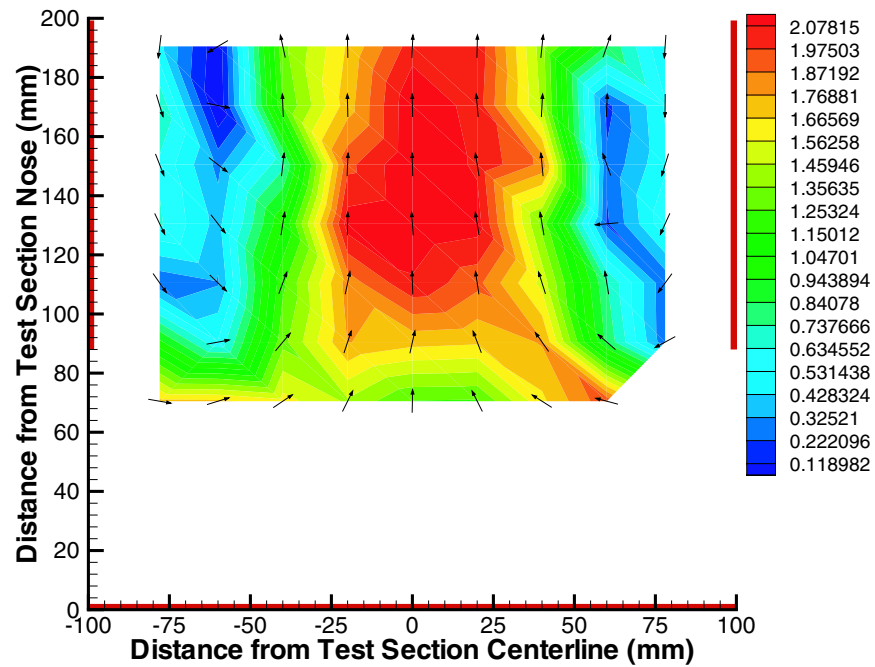


Figure A.21 Velocity measured in the nominal WTHL test section at 70 percent of the nominal flow rate at the test section centerline.

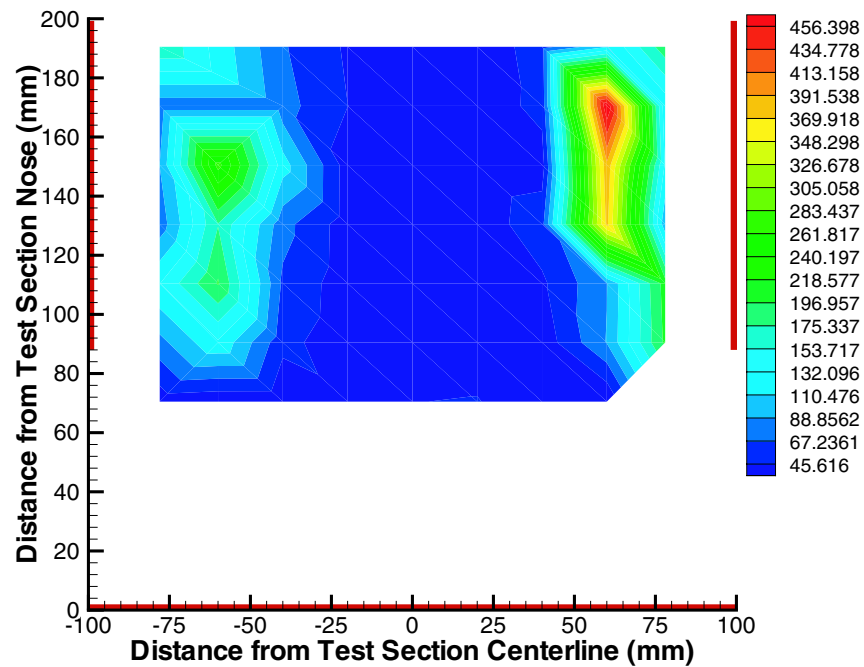


Figure A.22 Turbulence intensity measured in the nominal WTHL test section at 70 percent of the nominal flow rate at the test section centerline.

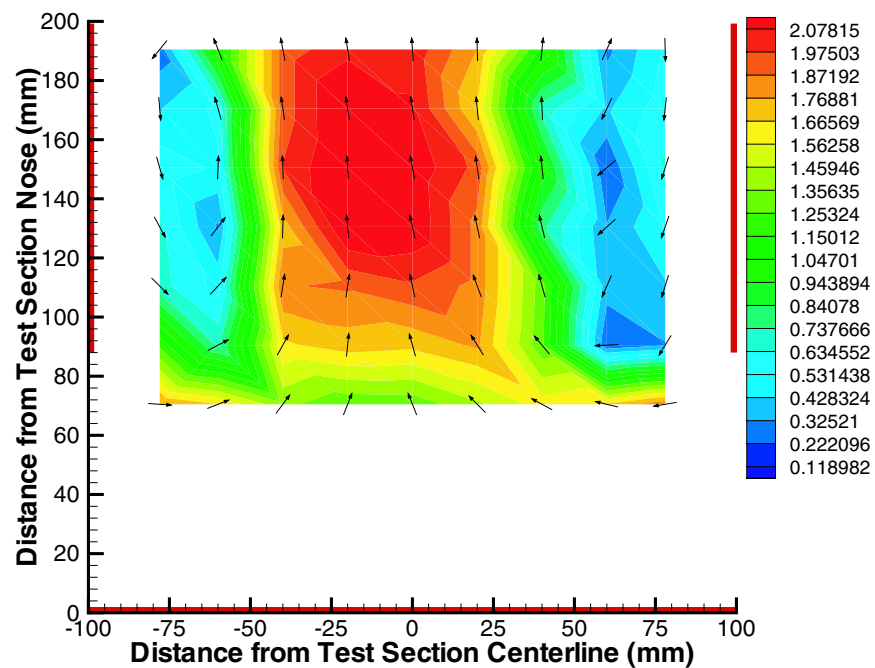


Figure A.23 Velocity measured in the nominal WTHL test section at 70 percent of the nominal flow rate at the test section centerline.

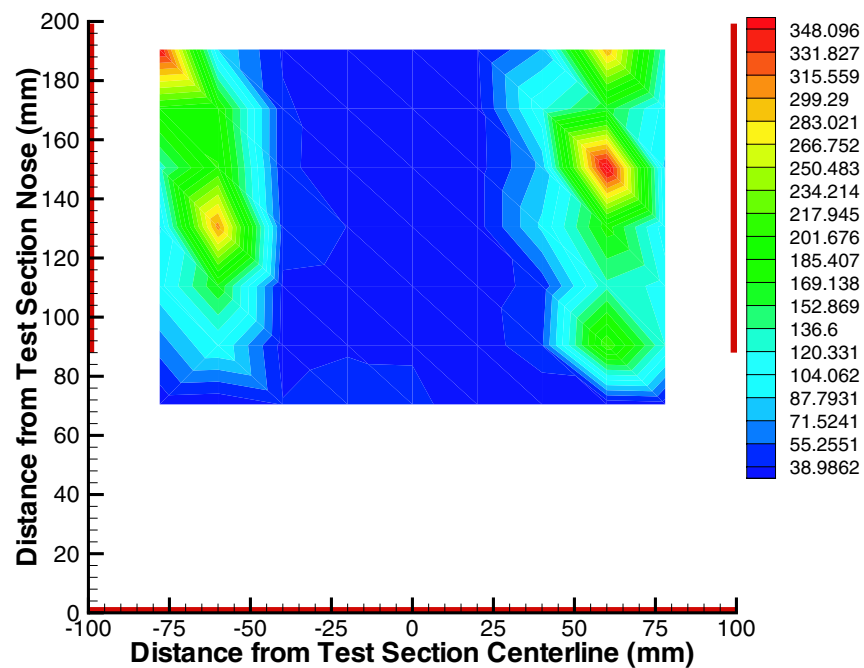


Figure A.24 Turbulence intensity measured in the nominal WTHL test section at 70 percent of the nominal flow rate at the test section centerline.

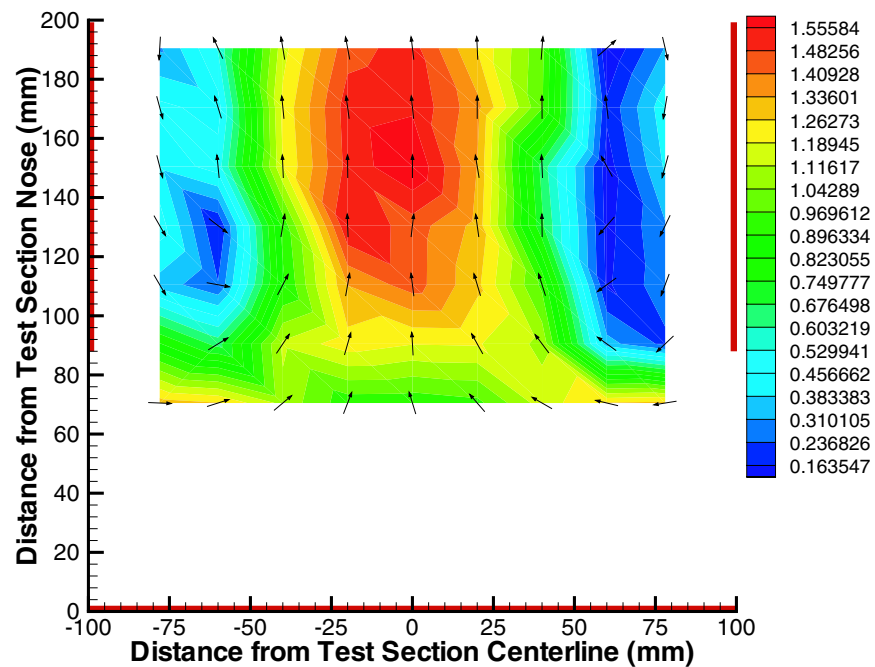


Figure A.25 Velocity measured in the nominal WTHL test section at 50 percent of the nominal flow rate at the test section centerline.

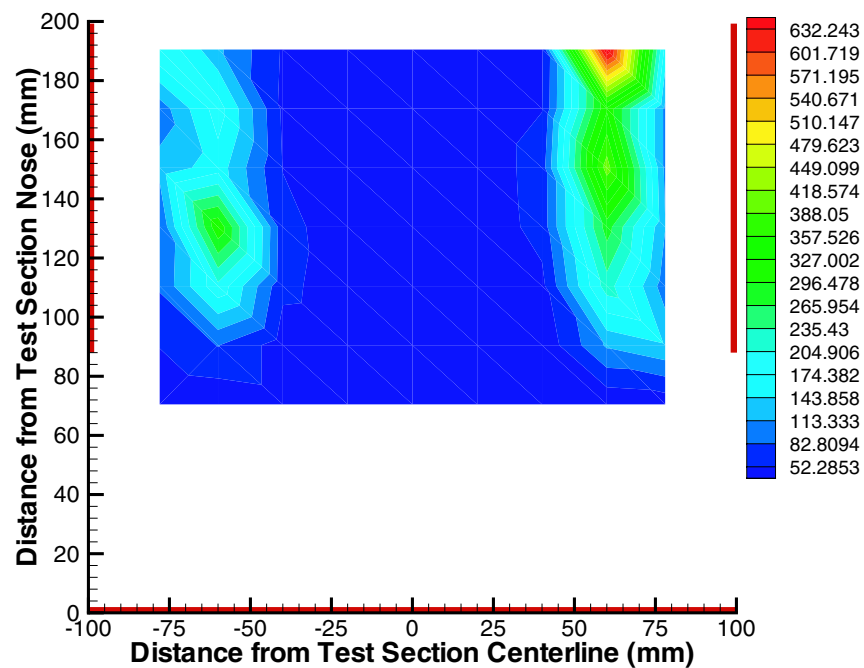


Figure A.26 Turbulence intensity measured in the nominal WTHL test section at 50 percent of the nominal flow rate at the test section centerline.

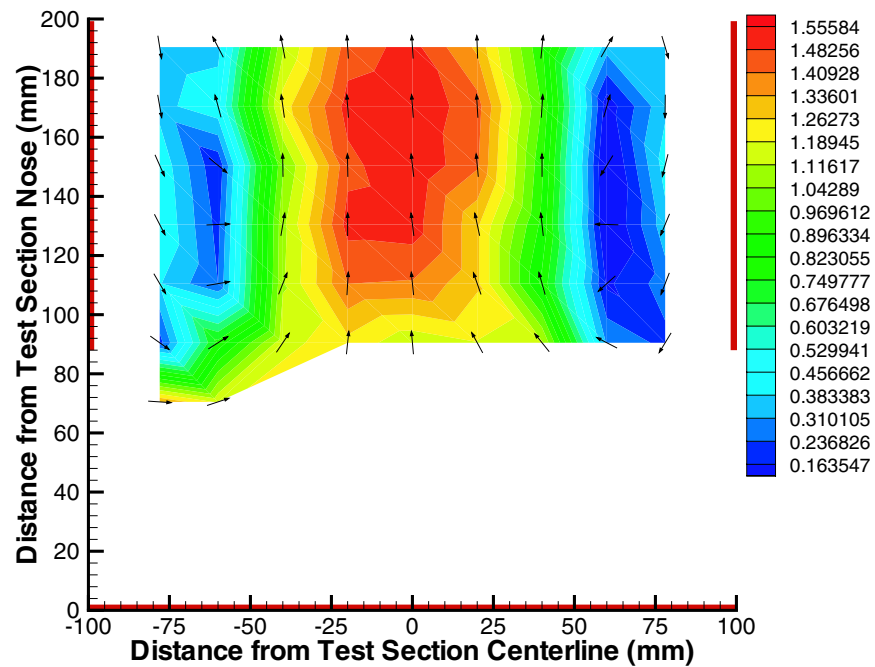


Figure A.27 Velocity measured in the nominal WTHL test section at 50 percent of the nominal flow rate at the test section centerline.

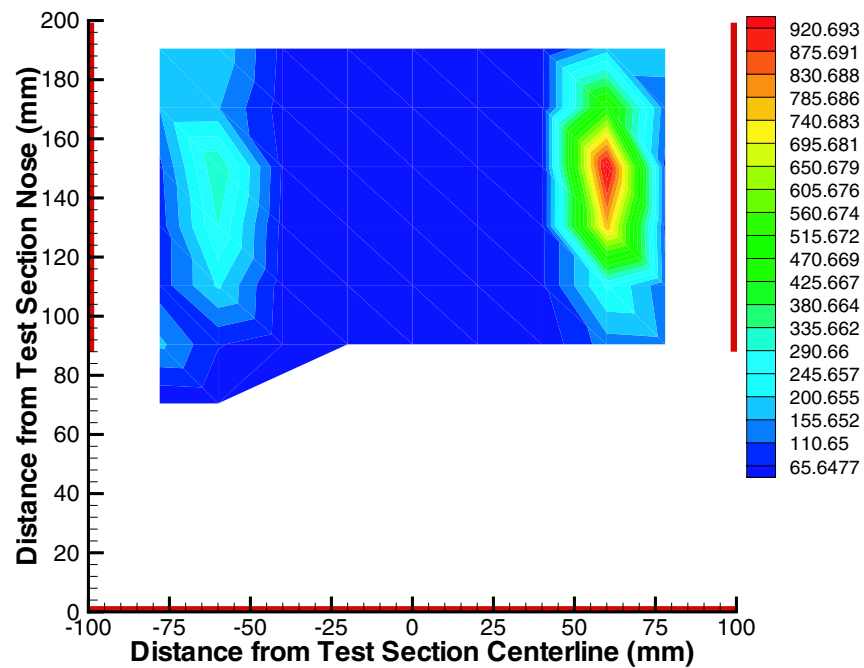


Figure A.28 Turbulence intensity measured in the nominal WTHL test section at 50 percent of the nominal flow rate at the test section centerline.

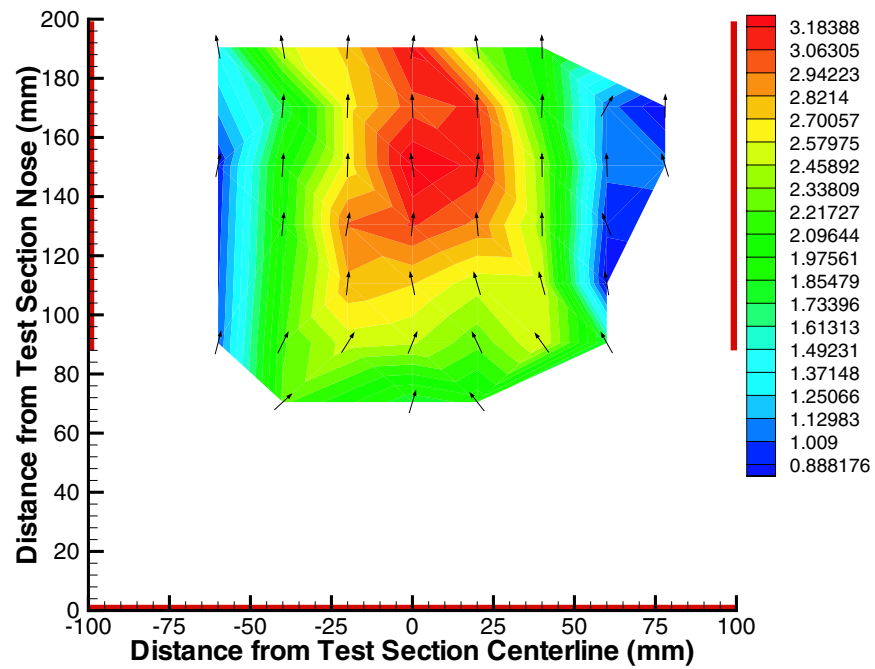


Figure A.29 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 20 mm from the bottom inside surface of the test section.

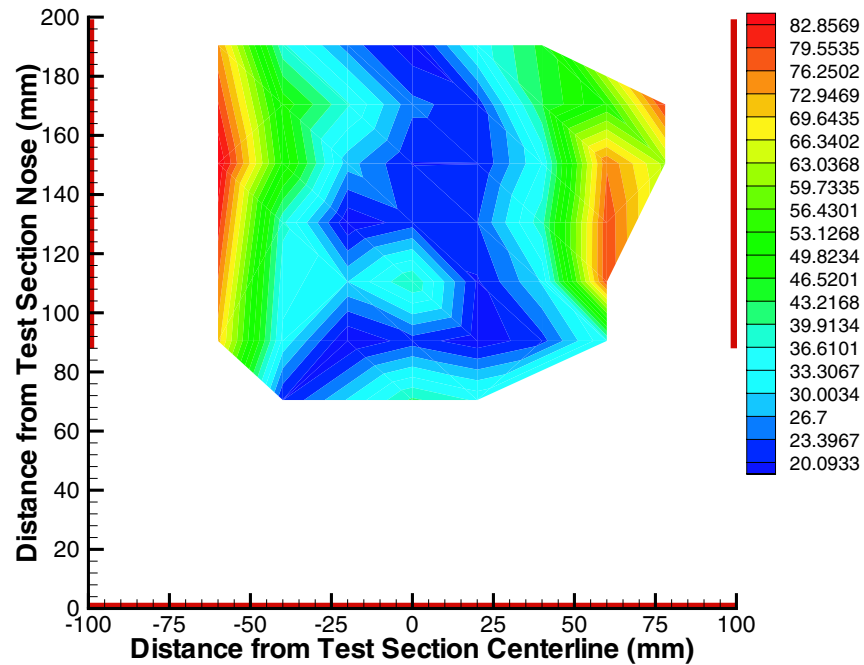


Figure A.30 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 20 mm from the bottom inside surface of the test section.

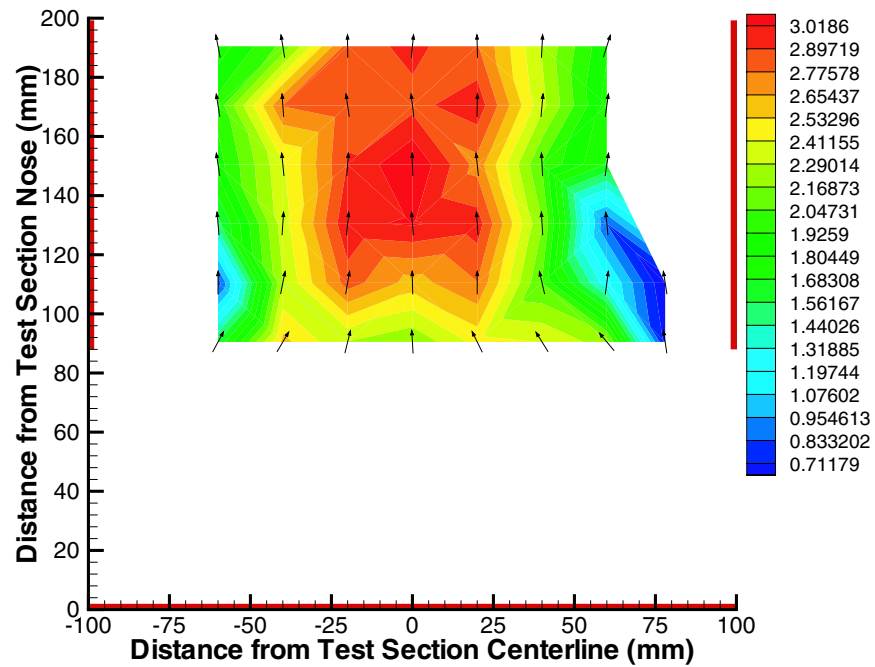


Figure A.31 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section.

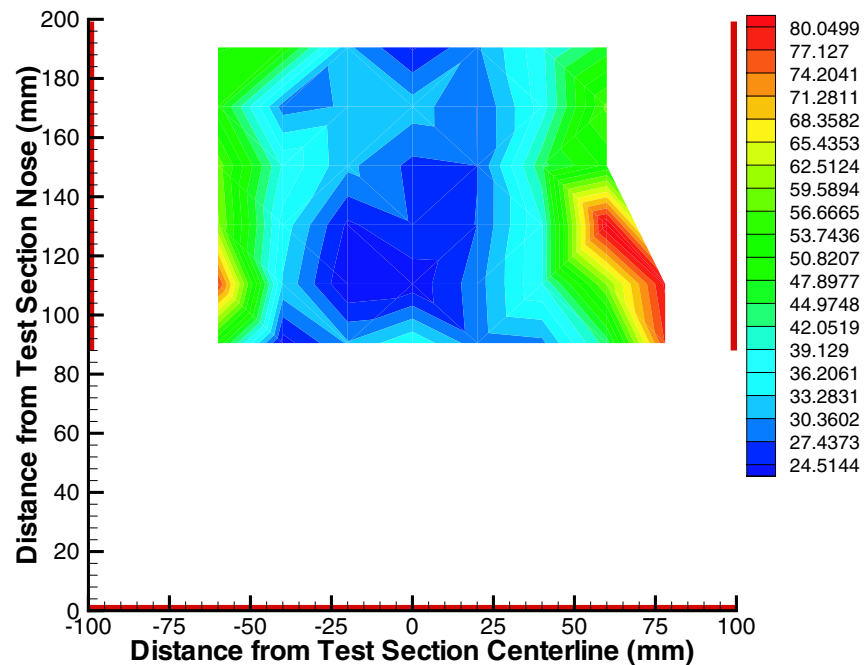


Figure A.32 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section.

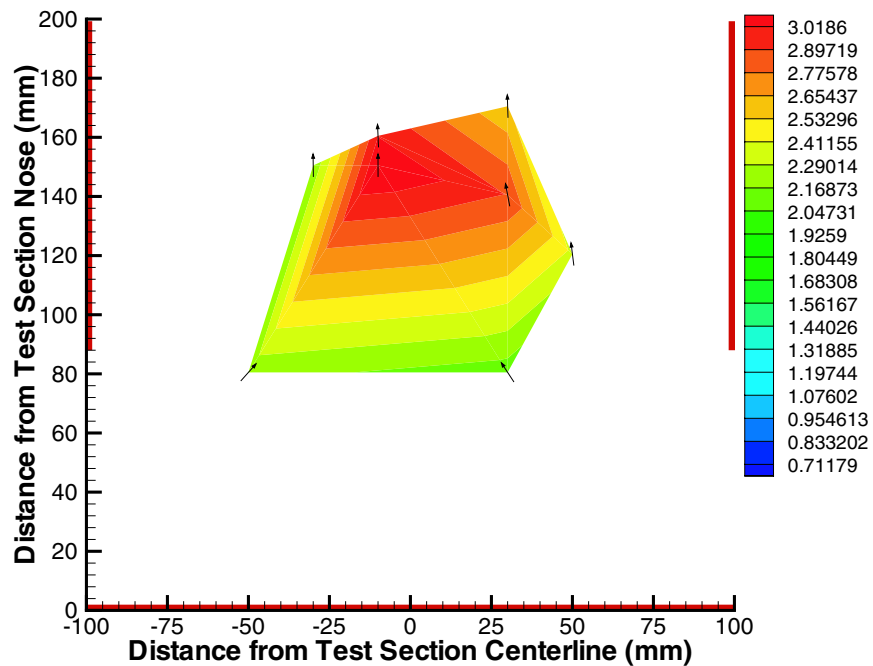


Figure A.33 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section. This data set is incomplete due to rust buildup in the loop process water.

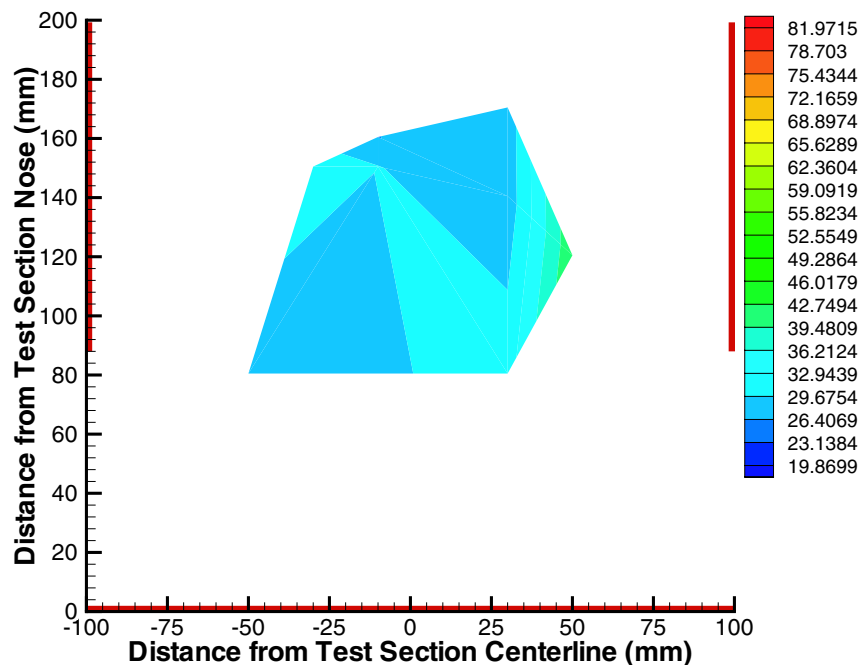


Figure A.34 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section. This data set is incomplete due to rust buildup in the loop process water.

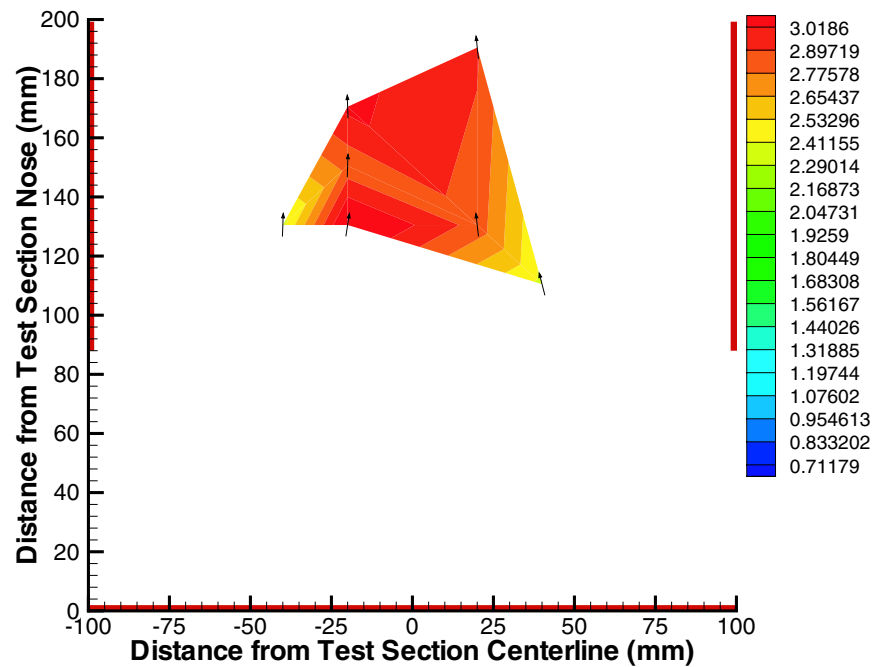


Figure A.35 Velocity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section. This data set is incomplete due to rust buildup in the loop process water.

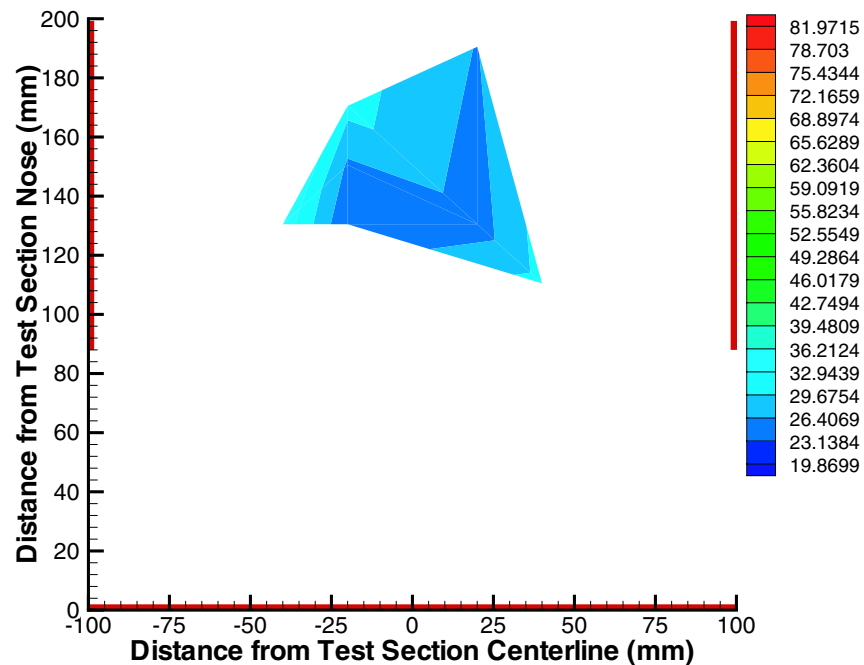


Figure A.36 Turbulence intensity measured in the nominal WTHL test section at nominal flow rate at a depth of 10 mm from the bottom inside surface of the test section. This data set is incomplete due to rust buildup in the loop process water.

Vita

William David Pointer was born in Knoxville, Tennessee on September 9, 1975. He attended elementary and secondary schools in the public system of Knox County, Tennessee, where he graduated from Karns High School in 1993. He entered The University of Tennessee in the fall of 1993, majoring in Nuclear Engineering. He obtained his Bachelor of Science degree in Nuclear Engineering in the spring of 1997. He obtained the Master of Science degree in Nuclear Engineering at the University of Tennessee in the summer of 1999 with the completion of the thesis titled "Evaluation of Low Pressure Leak Flows in Metallic Vacuum Vessels for Detection Systems Using Helium Dissolved in an External Water Cooling Jacket." He continued his studies at the University of Tennessee and began working toward a Doctor of Philosophy degree in the fall of 1999. He received his Ph.D. in Nuclear Engineering in the spring of 2001.

He joined the SNS program at Oak Ridge National Laboratory as a graduate student under contract through the university in the summer of 1999 and worked on a variety of engineering problems including radiation heat transfer in spent target vessels in the storage facility, scaling and similarity studies for the heat transfer experiments, and the experimental characterization of the flow patterns in the mercury target. While at the laboratory, he also worked on safety analyses of Russian VVER-1000 nuclear power plants burning MOX fuel as part of the Joint Russian-American Weapons Grade Plutonium Disposition Program.

He and his wife, Angela, are currently living in Chicago, Illinois, where he is working at the Argonne National Laboratory as part of the Reactor Analysis and Engineering Division. His research activities focus on the development of computational models for thermal hydraulic evaluations and development of scaled experiments for validation, verification and benchmarking.